Universidade da Coruña

PhD Dissertation

Topology optimization of structures with high spatial definition considering minimum weight and stress constraints

Diego Villalba Rama

Supervisors:

Dr. José París López
Dr. Fermín Navarrina Martínez

PhD in Civil Engineering
March 2021

Acknowledgments

As the old saying goes "No pain no gain". Although the development of a doctoral thesis is a laborious task, since the majority of the things that are done does not go well at the first attempt, the pleasure that supposes to collaborate with the humanity by contributing to the global knowledge compensates all the effort required and the bad moments spent during its development.

However, all this would not be possible without the support of the people who had surrounding me along all this journey. Therefore, and as another old saying goes "One good turn deserves another", it is time to be thankful to those people that have helped me out throughout this travel that has taken me several years.

First of all, I would like to express my special appreciation to my advisors: Dr. José París and Dr. Fermín Navarrina for their support and their advice during the development of this Thesis. This gratitude is extended to all the members of the Group of Numerical Methods in Engineering (GMNI) for their help, their advice and their support but in particular to Luis Ramírez and Iván Couceiro.

I would also like to express my gratitude to Javier Fernández, Manuel Ruíz, Antonio Eiris and Santiago Méndez for providing a really comfortable atmosphere not only in the Virion Break Lab. but also in the recesses.

On the other hand, I don't want to forget my friends who have supported me over the last years, not only during the development of this thesis but also during the studies of Civil Engineering: Alberto P., Andrea M., Pablo, Danilo, Fernando, María, Antía, Andrea T., Jorge, Alberto M., Alex, Cris, Emma y Yoli.

I would also like to thank my family for their support during the development of this thesis. Last but not least, I would like to express my sincere respect and gratitude for the effort my parents have made in order for me to be where and what I am now.

This work has been partially supported by the “Xunta de Galicia (Secretaría Xeral de Universidades)” and the European Union (European Social Fund - ESF) through “Grants for supporting the predoctoral stage in universities of the Galician University System, in research public organisms of Galicia and in other entities of the “R & D system of the Galician Government - 2016” ED481A-2016/387, by Feder funds of the European Union, by the “Ministerio de Economía y Competitividad” of the Spanish Government through grants DPI2015-68341-R and RTI2018-093366-B-I00, by the “Consellería de Educación, Universidade e Formación Profesional” of the “Xunta de Galicia” through “grants for the consolidation and structuring of competitive research units of the Galician University System: Competitive reference group” GRC2014/039 and GRC2018/41, and by research fellowships of the University of A Coruña and the “Fundación de la Ingeniería Civil de Galicia”.

Abstract

The first formulation of Topology Optimization was proposed in 1988. Since then, many contributions have been presented with the purpose of improving its efficiency and extending its applicability. In this thesis, a topology optimization algorithm that allows to obtain the structure of minimum weight that is able to support different loads is developed. For this purpose, the requirement that stresses have to be lower than a maximum value has been considered in its development.

Although the structural topology optimization problem with stress constraints have been previously formulated with several different approaches, a Damage Constraint approach is developed in this thesis to incorporate them in a different way. The main objective of this modification is to reduce the CPU time required in the solution of the topology optimization problem. This reduction will allow to solve problems with a higher number of design variables what enables the attainment of solutions with high spatial definition.

Moreover, two different approaches are used to define the material distribution in the domain: uniform density per element formulation and material density distribution by means of isogeometric interpolation. In the first approach the Finite Element Method (FEM) is used to solve the structural analysis and the relative density in each element of the mesh is chosen as design variable, while the second one uses the Isogeometric Analysis (IGA) for solving the structural analysis and the values of the relative density at a certain number of control points are used as design variables.

On the other hand, the optimization is addressed by using Sequential Linear Programming, that requires a first order sensitivity analysis. All the sensitivities are obtained through analytic derivatives by using both, direct differentiation and the adjoint variable method. Finally, some application examples are solved by means of both methods (FEM and IGA) in the two-dimensional and three-dimensional space.

Resumen

La primera formulación de la Optimización Topológica fue propuesta en 1988. Desde entonces muchas aportaciones se han presentado para mejorar su eficiencia y extender su aplicabilidad. En esta tesis se desarrolla un algoritmo de optimización topológica que permita obtener la estructura de mínimo peso que sea capaz de soportar diferentes cargas. Para este propósito se ha considerado en su desarrollo la condición de que las tensiones sean inferiores a un cierto valor máximo.

Aunque el problema de optimización topológica estructural con restricciones de tensión se formuló previamente con diferentes enfoques, en esta tesis se desarrolla un enfoque que considera una restricción de daño para incorporarlas de una forma diferente. El principal objetivo de esta modificación es reducir el tiempo de computación requerido en la solución del problema de optimización topológica. Esta reducción permitirá resolver problemas con un mayor número de variables de diseño lo que a su vez permite la obtención de soluciones con alta definición espacial.

Para definir la distribución de material en el dominio se usan dos formulaciones diferentes: formulación de densidad uniforme por elemento y distribución de material por medio de una interpolación isogeométrica. El primer planteamiento usa el Método de los Elementos Finitos (MEF) para resolver el análisis estructural y toma como variable de diseño el valor de la densidad relativa en cada elemento de la malla, mientras que el segundo requiere del uso del Análisis Isogeométrico (IGA) para resolver el análisis estructural y los valores de la densidad relativa en un cierto número de puntos de control son las variables de diseño.

El problema de optimización se resuelve con las técnicas de Programación Lineal Secuencial requiriendo únicamente el análisis de sensibilidad de primer orden. Todas las derivadas se calculan por derivación analítica haciendo uso de las técnicas de derivación directa y del método de la variable adjunta. Finalmente, se resuelven algunos ejemplos de aplicación con ambos métodos (MEF e IGA) en el espacio bidimensional y tridimensional.

Resumo

A primeira formulación da Optimización Topolóxica foi proposta en 1988. Desde entón moitas achegas se presentaron para mellorar a súa eficiencia e estender a súa aplicabilidade. Nesta tese desenvólvese un algoritmo de optimización topolóxica que permita obter a estrutura de mínimo peso que sexa capaz de soportar diferentes cargas. Para este propósito considerouse no seu desenvolvemento a condición de que as tensións sexan inferiores a un certo valor máximo.

Aínda que o problema de optimización topolóxica estrutural con restricións de tensión formulouse previamente con diferentes enfoques, nesta tese desenvólvese un enfoque que considera unha restrición de dano para incorporalas dunha forma diferente. O principal obxectivo desta modificación é reducir o tempo de computación requirido na solución do problema de optimización topolóxica. Esta redución permitirá resolver problemas cun maior número de variables de deseño o que á súa vez permite a obtención de solucións con alta definición espacial.

Para definir a distribución de material no dominio úsanse dúas formulacións diferentes: formulación de densidade uniforme por elemento e distribución de material por medio dunha interpolación isoxeométrica. A primeira formulación usa o Método dos Elementos Finitos (MEF) para resolver a análise estrutural e toma coma variable de deseño o valor da densidade relativa en cada elemento da malla, mentres que o segundo require do uso da Análise Isoxeométrica (IGA) para resolver a análise estrutural e os valores da densidade relativa nun certo número de puntos de control son as variables de deseño.

O problema de optimización resólvese coas técnicas de Programación Lineal Secuencial requirindo unicamente a análise de sensibilidade de primeira orde. Todas as derivadas calcúlanse por derivación analítica facendo uso das técnicas de derivación directa e do método da variable adxunta. Finalmente, resólvense algúns exemplos de aplicación con ámbolos métodos (MEF e IGA) no espazo bidimensional e tridimensional.

Contents

List of Figures

List of Tables

Prologue

"To write well, express yourself like the common people, but think like a wise man"
Aristotle

Once upon a time a child worried about the future of his planet. The levels of carbon dioxide in the atmosphere were extremely high, the sea level started to rise worryingly and the natural resources were about to run out. However, it existed a little possibility to change this situation and to return the planet to its balanced situation.

Although most of the prime ministers do not have any idea about the strategies that they have to do to save the planet. This child and other like him, started to share their ideas about the actions that all the people should do to protect the future of humanity. Most of them thought that the most polluted industry had to be closed, however, this child believed that they could maintain their lifestyle if the industries produced their goods in a more efficient way.

The years passed and the child became a teenager. Once he passed his entrance examination, he started his engineering degree with the objective of realizing his childhood believes. Fortunately, he finished his engineering degree and he decided to make a doctoral thesis, since he wanted to develop a way to define structures, components or elements that would require the minimal amount of natural resources and electric power during its manufacture and would be able to develop the task for what it was designed. Regrettably, we can not know if this man achieved this objective, since this is the beginning of the thesis that he wrote.

Although the writing of a PhD thesis is considered by many to be an impossible challenge, people who have already written it believe that the challenge of reading it is considerably higher, since it can be possible that you - the reader - may find the theoretical developments tedious and intricate. On the other side, you might be particularly interested mostly in a certain part of the thesis, what may lead you to ignore the rest. Therefore, the writer is going to do his best and will try to attract your attention towards reading all the document.

To conclude, the writer hopes that you will find this dissertation useful and that you are interested in knowing if he has finally achieved his objective.

"Learning is the only thing the mind never exhausts, never fears and never regrets"

Leonardo da Vinci

Chapter 1

Introduction

"If a man will begin with certainties, he shall end in doubts, but if he will content to begin with doubts, he shall end in certainties"
Francis Bacon, (1561-1626).

1.1. Introduction

Since the beginning of mankind, man has been always interested in designing and building both structures and tools. These achievements can suppose not only an improvement of their living conditions but also a demonstration of their primacy over the rest of the humanity, since throughout the history, there has been a competition among all the historical civilizations or current countries to dominate the world.

Although initially the first progresses were possible due to the observation of the nature. The evolution of the human being, what generally supposes the development of new skills such as logical thinking, made the use of experimental test possible. As a result of this, trial-error techniques started to be used in order to design new structures or tools.

However, it took a long time until the man was able to design structures and tools without the need of observing the world that surrounded them. The comprehension of the natural phenomena meant that the human being was progressively acquiring its knowledge in different areas such as mathematics or physics. For this reason, the evolutionary process has been extremely slow, until the last century.

In the process of creating, there are two main phases. The first is designing. In this phase several aspects have to be established, such as its shape, its size or its material. The second is manufacturing. For this purpose, only the available techniques with respect to the machinery and the manual labor can be used.

The two main historical events that have supposed an important improvement in the design and manufacturing phases, especially in the case of tools, are the Industrial

Revolution that took place in the 18th century and the Computational Revolution in the 20th century. On the one hand, the Industrial Revolution produced an important improvement in the production phase due to the use of machines in the manufacturing of structures and tools. On the other hand, the Computational Revolution made possible the use of machines in the design phase, what supposed an important progress, since computers are capable of doing the same calculations as humans, but much quicker.

In spite of these events, all these improvements had not been possible without the development of all mathematical theories that make possible the use of computers in order to perform the complex calculations that are required during the design phase. As a result of the Computational Revolution a big amount of calculation tools has been developed and, therefore, formulations that are based on different theoretical studies have been proposed.

Even though these techniques were initially used only to check the validity of the solutions that had been previously designed, a growing interest in the attainment of the optimum solution made necessary the development of new techniques that were also able to modify the initially proposed design. As a consequence, optimization techniques were developed.

Nevertheless, the optimization techniques did not suppose an immediate improvement, since an iterative procedure is required. Thus, a certain number of changes must be made between two consecutive iterations and all possible solutions have to be tested. Because of this, it was necessary to develop algorithms that could be able to predict the changes that have to be done with regard to the solution in a better way. This also produces a reduction in the amount of time required to solve the problem.

Nowadays, a big effort is being made in order to improve the efficiency of the optimization techniques. This interest could be exclusively due to the desire of achieving a big economic profit in some cases, although a reduction of the impact that the manufacturing process has over the environment is also obtained at the same time.

At this point, it is important to take the biunivocal correspondence between the design phase and the manufacturing phase into consideration. At first, the solutions that were provided by the optimization techniques, were technically unattainable, because of their complex geometry, however, the recent development of three dimensional printing techniques makes possible the manufacturing of almost whatever solution could be imagined.

On the other hand, the three dimensional printing industry can make possible in the near future the reuse of pollutant materials such as plastic in the manufacturing of tools or other elements, or concrete in the case of structures.

However, in this moment, the situation is not satisfactory since the main problem is that, notwithstanding the above, three-dimensional topology optimization techniques have not been extensively developed yet. Among other reasons, the techniques and algorithms that are being used in two-dimensional topology optimization can not be directly implemented in three-dimensional problems due to the computational cost that is required to achieve high spatial definition.

As a consequence of this, the main effort today is made in the development of new techniques that could allow for a decrease in the amount of time that has to be used in the solution of topology optimization problems of three-dimensional components or structures.

1.2. Evolution of the structural topology optimization

Although the optimization of structures had been developed many years ago, the way this issue had been initially formulated limited its area of application to a certain kind of problems. The main drawback of the former approaches was the high responsibility of the designer in the choice of the structural topology. Obviously, this decision has a determining influence in the final solution that will be obtained.

Therefore, all this means that for the same structural optimization problem, several different solutions will be obtained depending on, among other things, the preferences of the designer with respect to the structural topology. However, it is obvious that the solution of each structural optimization problem should be unique. In other words, the final solution to the structural optimization problem should be the same independently on the decisions taken along the optimization process. All this called for the developments of the first topology optimization formulations.

The first approach of the topology optimization problem was proposed by Bendsøe and Kikuchi [Bendsøe & Kikuchi, 1988] in 1988. The contributions that have been made in this field since then, especially in the solution of two-dimensional problems, have been extremely numerous with respect to not only the formulation of the problem, but also to their areas of application. However, the number of scientific publications devoted to three-dimensional problems is much lower than in the case of two-dimensional problems, due to, over all, the significant difference between the implied computational requirements.

The topology optimization problem was usually formulated by means of two different approaches, that will be described below. Its scope of application is extremely wide, including, inter alia, civil, aerospace, industrial engineering, or even in medicine.

Most of the formulations of topology optimization of structures had as an objective, the material layout in a certain domain in order to maximize or to minimize a certain characteristic. Maximizing the structural stiffness —or, in other words, minimizing the structural compliance— while imposing a constraint on the maximum amount of material that can be used was initially the way to formulate the problem.

This kind of formulation did not cause as interest as those approaches that take into account stresses or displacements as constraints in the problem. The main reason is that the stiffest structure is hardly ever the cheapest and the most efficient one, because the main objective is to ensure that the solution that will be obtained will be able to develop the role for what it was designed. Moreover, in all structural problems structural stresses and displacements are always relevant, while the same can not be said for stiffness.

Ever since Bendsøe and Kikuchi established the basis of the topology optimization, the majority of the research work has been focused on the solution of maximum stiffness problems. The implementation of volume constraints is much easier than the implementation of other kind of constraints, such as stresses or displacements. This is the main reason whereby this approach has been historically preferred.

However, from an engineering point of view, minimum weight formulations are much more practical. The main reasons are that not only the cheapest solution is obtained, but it is also possible to ensure that the obtained solutions will be able to resist the load cases that can be applied during the service life. This is possible because the objective function of the minimum weight problem has to do with the structural weight while the constraints take care of the verification of stresses and displacements.

Apart from this limited applicability, maximum stiffness formulations have some concomitant problems. Checkerboard solutions can be obtained in some cases. The obtained solutions exhibit a big dependency on the mesh that is used for the analysis of the structural problem, what is known as mesh dependency. Furthermore, these phenomena and the techniques used to solve these problems can be observed in [Bendsøe, 1995] and [Eschenauer & Olhoff, 2001].

As a consequence of the limitations of the maximum stiffness formulation other alternative approaches, such as minimum weight with stress constraints, have been developed. The formulation of minimum weight with stress constraints avoid some of the typical disadvantages of the maximum stiffness approach, such as the appearance of checkerboard configurations and the uncertainty on the feasibility of the obtained solutions. Even so, the mesh dependency remains being an intrinsic problem of the formulation.

Despite the name of the approach, stress constraints can be complemented or replaced by other kind of constraints, such as displacement or strain constraints. Moreover, as mentioned above, it is possible to consider several load cases in the solution of the topology optimization problem. Nonetheless, another important advantage of this approach from both, the engineering and the industrial points of view, lies in the formulation of the objective function. In this kind of problems, this function is the structural weight that is related with the manufacturing cost.

Although, minimum weight with stress constraints formulations aroused a big interest at first. This approach has also its own advantages and drawbacks. In the first developments of this procedure, each stress that has to be checked had its own constraint. In other words, each stress constraint has to be formulated separately. However, this way to impose stress constraints means a big handicap, as the number of constraints, that is related to the number of design variables that are used in the definition of the material layout, is heavily increased.

As a result of this increase in the number of stress constraints that have to be imposed, new ways to formulate the stress constraints in the topology optimization were developed. These formulations try to take into account a high number of stresses with a reduced number of constraints. As a consequence of that, a significant reduction in the computational requirements will be achieved.

For this purpose, formulas that make an approximated calculation of the maximum value of a certain set of values are used as a way to impose the stress constraints. In this case, this set of values will be a part or the whole stresses that have to be verified. These new kind of constraints are intended to check that the maximum value of the stresses provided by the formula is lower than the maximum allowable value. These ways to introduce the stress constraints in the topology optimization problem are known as aggregation stress constraints techniques.

Even though, the results that had been obtained with the aggregation stress constraints techniques are good enough, these methods also have some disadvantages. In some cases, the value of the stress at certain points can be higher than the maximum allowable value. This situation goes unnoticed since the procedure ignores these facts. However, the violation of the stress value would be known if a feasibility analysis on the value of the stress would be made. Despite of this, it would not be possible to solve this problem unless the way to define the constraint or the number of stresses considered in each group is changed.

With the aim of overcoming these drawbacks, a different strategy is developed in this Thesis to combine the stress constraints. This method will have to be able to ensure that all stresses will be lower than their maximum allowable value. With respect to the topology optimization problem of structures, the objective function will be the structural weight. The constraints will be the damage constraint and the side constraints. This damage constraint will evaluate the possible unfeasibility of the structural stresses.

On the other hand, the other important aspect that has to be taken into account in the formulation of the topology optimization problem is the way density will be defined over the design domain. Just like the constraints of the topology optimization problem, the definition of the density in the domain has also evolved over time.

In the first formulations of the topology optimization problem, a discrete definition of the material layout was used. In other words, there are only two possibilities in each point of the domain: material or void. The solutions obtained with this formulation are known as full-void solutions, and they are conditioned by the definition of the mesh. Moreover, this strategy in the definition of the density does not allow the use of Mathematical Programming algorithms. And in consequence, all the possible solutions have to be explored in order to obtain the optimal solution of the problem.

Due to these limitations of discrete formulations, a continuous formulation of the material layout is developed. In this case, and just as in discrete formulations, the domain is divided in a certain number of areas. Now, the density can take whatever value between a minimum and a maximum that are associated to this regions being void or full of material, respectively. As a consequence, the values of density can be used as design variables. In spite of this, the definition of the mesh continues being a problem.

The next strategy that has been developed in order to try to mitigate the mesh dependency, lies on the definition of the material layout in the domain in terms of a certain set of density values, that play the role of design variables, and a set of shape functions that determine the contribution of each design variable at every point of the domain. With this method, the value of the density will be continuous in all the domain, although the effect of mesh dependency will be also present. Finally, other approaches have been developed with the aim of precluding the mesh dependency problems induced by the definition of the material layout.

1.3. Objectives

The main objective of this thesis is the development of a method to solve topology optimization problems of structures on the basis of the weight minimization approach and the indirect incorporation of stress constraints by means of a damage constraint formulation, not only in two-dimensional cases, but also in three-dimensional ones. During the attainment of an efficient algorithm, it will be necessary to pay special attention to the theoretical formulation aspects, the numerical implementation of the algorithms and the computational requirements, especially with respect to the computing time. However, this main objective can be separated in other more specific:

- First of all, to make a study of the state of the art. The main objective is to have the maximum possible information about the ways topology optimization had been formulated. Furthermore, an analysis of the techniques that had been developed in the attainment of the solution with respect to the imposition of constraints and the definition of the material layout will be made.

- To make an extent of the two-dimensional approach of the structural analysis, based on Finite Element Method formulations to the three-dimensional space. This approach has been developed by the advisors of this thesis, however, the consideration of material layout in the domain will be also done.

- To incorporate the Isogeometric Analysis in the algorithms, that has been previously developed, as an alternative to the Finite Element Method. This incorporation makes possible the attainment of solutions with a high spatial definition by means of the use of less elements in the definition of the problem.

- To develop the formulation of the topology optimization problem of structures with minimum weight and an indirect incorporation of the stress constraints. In this point, it will be also necessary to make an analysis of the objective function and establish the best way to introduce a constraint that takes all structural stresses into consideration.

- To analyze the limitations of the method proposed. If it would be possible to propose a set of improvements to reduce the amount of time that is required in the solution of the topology optimization problem. Some clarifications are introduced in order to make the method more easily understandable.

- To analyze all the optimization algorithms that have been previously developed. This analysis will let choose the most efficient of them. The objective is to solve the topology optimization problem in an appropriate way and in a little amount of time.

- To make the implementation of the algorithms that have been chosen previously, by means of programming techniques in order to solve topology optimization problems, not only in two or three dimensions, but also with Finite Element Method or Isogeometric Analysis.

- To make a selection of typical topology optimization problems in two dimensions that had been solved theoretically and by means of previous topology optimization algorithms in order to check the operation of the algorithm developed in this thesis through the comparison of the solutions. Once the feasibility is validated more practical examples in two dimensions are solved.

- To make a search of topology optimization problems that have been solved in three dimensions. These problems will be solved in order to test the operation of the algorithm in the three-dimensional space, by means of the comparison between both solutions. The time that have to be used in the attainment of the solution will let analyze its efficiency.

1.4. Structure of the thesis

This dissertation is composed by 10 chapters, including the introduction one. In addition, this document includes two appendixes and the bibliography.

In this first chapter, a review of the historic events that had influence in the design and manufacturing phase is made. Then the evolution of the topology optimization of structures is presented, paying special attention to all the approaches that had been developed until now in order to solve the problem. Finally, the main objectives of this thesis are established.

In the second chapter, a review of the state of the art is presented. In the first place, the most relevant methods that have been used to formulate the problem in terms of the material density as a design variable are described. Then the different formulations that have been used to impose stress constraints in order to solve the topology optimization problem are commented. Finally, the concept of damage approach as a way to impose constraints is introduced.

In the third chapter, the basic concepts of the isogeometric analysis are introduced, such as: knot vector, basis function and B-splines. Then, the way isogeometric analysis can be implemented in a finite element method code is analyzed. Finally, the B-spline surfaces or solids that are used in the rest of the procedure are developed.

In the fourth chapter, the formulation of the structural analysis with Finite Element Method is developed. Then the considerations that have to be taken into account in the structural analysis when the Isogeometric Analysis is incorporated are analyzed.

Finally, the influence that material layout in the domain has in the formulations is taken into consideration.

In the fifth chapter, the formulation of the topology optimization problem of structures with minimum weight and stress constraints is developed. First the objective function is defined for both methods: Finite Element Method and Isogeometric Analysis. Then, the concept of densities penalty is introduced. Thereupon, the damage constraint is completely formulated. Finally, the stress criterion that is introduced in the damage constraint and its singularity phenomena are commented.

In the sixth chapter, the optimization algorithms that are used in the solution of the topology optimization problem are developed. These algorithms depend on the results that had been obtained in the structural analysis and in the definition of the damage constraint. Then, once the improvement direction is obtained, the calculation of the improvement factor is established. For this purpose, the side constraints are taken into account.

In the seventh chapter, the sensitivity analysis is developed. First of all, a set of considerations that have to be taken into account are made. These considerations depend on the method that had been chosen to solve the problem. In the second place, the different derivation techniques used are developed. Finally, the sensitivity analysis is formulated not only for the objective function, but also for the damage constraint.

In the eighth chapter, the complete methodology of the topology optimization algorithm is developed. Then, the most important computational aspects in the numerical implementation of the topology optimization algorithm are established. Finally, the evolutionary parameters that are introduced in the algorithm in order to reduce the number of required iterations are analyzed.

In the ninth chapter, the numerical examples that have been solved are presented. First, 2D testing examples with Finite Element Method are presented in order to validate the topology optimization of structures with minimum weight and damage constraint. Then, 2D examples with Finite Element Method and Isogeometric Analysis are presented in order to check the feasibility of the Isogeometric Analysis as alternative to the Finite Element Method. Next, 3D examples with Finite Element Method and Isogeometric Analysis are presented. Finally, a comparative analysis between the use of Finite Element Method and Isogeometric Analysis about CPU time and computational requirements is made.

In the tenth chapter, the main conclusions about the thesis are commented. Moreover, the future research areas that have arisen as a result of this thesis are proposed.

On the other hand, the appendixes included in this document contain an overview of the contents of this thesis written in Spanish and in Galician, as required by the regulations of this University for the dissertations written in other languages.

Finally, the bibliography that has been indexed throughout the thesis is incorporated, alphabetically ordered.

Chapter 2

State of the art

"Study the past if you would define the future"
Confucius, (551BC-479BC).

2.1. Introduction

Ever since in 1988, Bendsøe and Kikuchi [Bendsøe & Kikuchi, 1988] submitted the first works about structural topology optimization and, consequently, stated the basis of this new field, an important number of contributions in this area has been made. Despite the fact that an important number of different kind of problems have been formulated, only two of them have been deeply analyzed, on the one hand, the maximum stiffness problem and on the other hand with a more recent development, the minimum weight with stress constraints problem. Furthermore, the majority of the works that have been published until now, with the exception of the maximum stiffness problem, have been developed in the two-dimensional space, because of the much larger computational requirements that have to be used in the solution of three-dimensional problems.

Although, as mentioned above, solely two kind of formulations have been widely studied, the variety of strategies developed to solve these problems are extremely high owing to the drawbacks that had emerged during the solution process. On the one hand, there are problems that are concomitant of a certain kind of formulation as the attainment of checkerboard solutions when maximum stiffness problems are solved. On the other hand, there are drawbacks that are common in both methods like the mesh dependency of the solutions. Finally, the desire of achieving high quality solutions introduces disadvantages related with the computational requirements that is associated with the CPU time. Nevertheless, the attention in this thesis will be only focused in the minimum weight with stress constraints approach because of its high interest from the engineering point of view.

First, the way material layout in the domain is defined has been widely studied. This circumstance will have a big influence in the appearance of the mesh dependency phenomenon, especially if a certain kind of structure is considered in its definition, because of the number of elements in that the domain is divided. For this reason, the different strategies that had been used to mitigate the mesh dependency will be analyzed in the following sections.

Then, the different approaches that have been used in order to impose the stress constraints will be also developed. This part of the problem is extremely important since it is the part that usually requires the majority of the computational requirements and CPU time and one of the main objectives of the topology optimization is the attainment of the optimal solution in a reasonable amount of time. Additionally, it is important to take into consideration that the number of stress constraints has to be at least equal to the number of design variables that are used in the definition of the material layout, if they are imposed individually. As a result of the desire of diminishing the CPU time, the stress constraint aggregation techniques had been developed.

Apart from that, three new strategies recently developed will be introduced: Multiresolution Techniques, Topological Derivative and Damage Approach. The first approach states the structural analysis and the material layout by means of different patterns in order to reduce the time consumption keeping the quality of the results. The second approach analyses the influence that the introduction of small holes (topology changes) in the structural domain produces in a certain characteristic, such as structural displacements. The third approach uses an alternative model that is perturbed when the constraints are violated. The global damage constraint will be imposed through the comparison between both models: the original one and the perturbed.

Finally, a brief overview about all the aspects that will be developed in this chapter and that, in some cases, will have to be used in this thesis will be briefly commented.

2.2. Structural weight definition: discontinuous formulations

Topology optimization problems were initially formulated by means of the use of discrete design variables, what usually means that the domain has to be divided in a certain set of areas, and each part of the domain can be just full of material or void. However, solving this kind of problems is extremely cumbersome, since design variables are discrete and considering all the possible combinations would require unaffordable computing requirements.

As a result of this, continuous design variables start to be used instead of discrete ones in the solution of the topology optimization problem. In spite of this, the attainment of full-void solutions continued being one of the main objectives because of their simplicity in the manufacturing procedure.

Nevertheless, the continuity concept can be related to two different aspects, not only the values that density can take at each point, but also the continuity in the density value throughout all the domain. The first aspect has been introduced previously and the second one has not been explained yet. Globally, continuous formulations of the density will be developed in the next section.

In this section, the different approaches for discontinuous formulations of the density in the domain will be introduced. In all this kind of methods and part of the continuous formulation approaches, the domain will have to be always divided in a certain number of areas, and in each region the value of the density or, in other words, the design variable will be constant. The partition of the domain in a certain number of areas is the cause of the appearance of the mesh dependency phenomenon, because if the number of areas is modified, the solution of the problem can be also modified.

2.2.1. Homogenization

Although, at first, the discontinuous formulation of the density did not suppose an important problem with respect to the constituent model of the material, since there were only two possibilities: full or void. The incorporation of intermediate values of density introduced some numerical and conceptual difficulties. First of all, it was necessary to define constituent equations for the intermediate values of the design variables, since this definition will be essential for solving the structural analysis.

As a result of this, a theory that can obtain the constituent model of whatever material with continuous design variables was developed. The most common technique to obtain this constituent model was by means of the definition of a robust microstructure that includes the effect of the design variable. As of this microscopic configuration and with homogenization techniques, it was possible to obtain the stiffness matrix at a macroscopic level. This matrix defines the structural behavior for intermediate values of the design variables.

This way of obtaining the constituent model is known as Homogenization Techniques. The basis of this approach was established before its application in the topology optimization field, by [Murat & Tartar, 1985]. Moreover, other contributions were also made in this field before the development of the basis of topology optimization, such as [Kohn & Strang, 1986a], [Kohn & Strang, 1986b], [Kohn & Strang, 1986c] and [Lurie & Cherkaev, 1997].

Once the basis of the topology optimization was established by [Bendsøe & Kikuchi, 1988], homogenization techniques experimented an important evolution, because of their easy applicability in the formulation of the Topology Optimization problem. However, homogenization techniques continue nowadays being one of the most used methods in order to establish the way material is distributed in the domain.

Some of the most important publications in this field had been [Allaire et al., 2004], [Bendsøe, 1995], [Eschenauer & Olhoff, 2001] and [Suzuki & Kikuchi, 1991] among others. On the other hand, the most common approaches of homogenization techniques that have been developed since the formulation of the basis of topology optimization will be commented in the next sections.

Hole-in-cell

One of the first kind of microstructures that have been used in the solution of the topology optimization problem of structures was the approach that is known as hole-in-cell method that was firstly proposed by [Murat & Tartar, 1985]. In this method, a square hole is introduced in each element of the mesh. Moreover, it is possible to orient this hole in whatever direction and its dimensions can take any value. However, the square hole was replaced by rectangular one what supposes a more general approach.

As a result of this generalization of the problem, its complexity was increased, because the number of design variables that have to be used was higher in comparison with the square holes. The design variables that have to be taken into account were the dimensions of the hole and its orientation. Finally, a general graphic representation of this approach can be observed in 2.1.

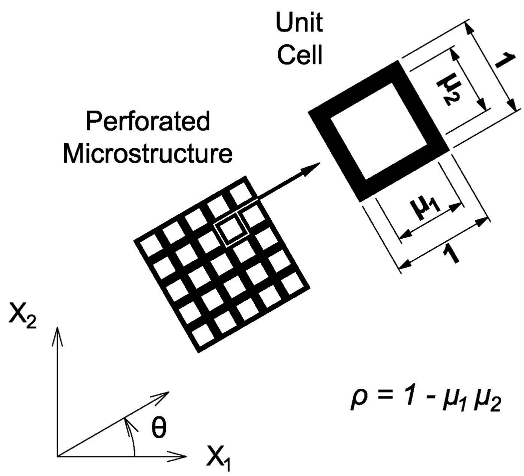


Figure 2.1. General hole-in-cell structure [Olhoff & Eschenauer, 1999]

Layered microstructures

Apart from the hole-in-cell approach, the layered microstructures approach was another kind of microstructures that has been widely used by the homogenization techniques. In this method, a certain number of layers of material with different properties and orientations were disposed. As a result of this, it is possible to classify these microstructures in different groups. Each of this groups had a certain number of layers whose fibers were oriented in a given direction.

On the other hand, the use of this method did not suppose any advantage with respect to the hole-in-cell approach, since the number of parameters that were necessary to define the microstructure continued being too big. In this case, the design variables were the orientation of the fibers, the number of layers that were necessary (usually equal to the number of dimensions) and the separation distance between fibers. Lastly, a general graphic representation of this approach can be watched in figures 2.2 and 2.3.

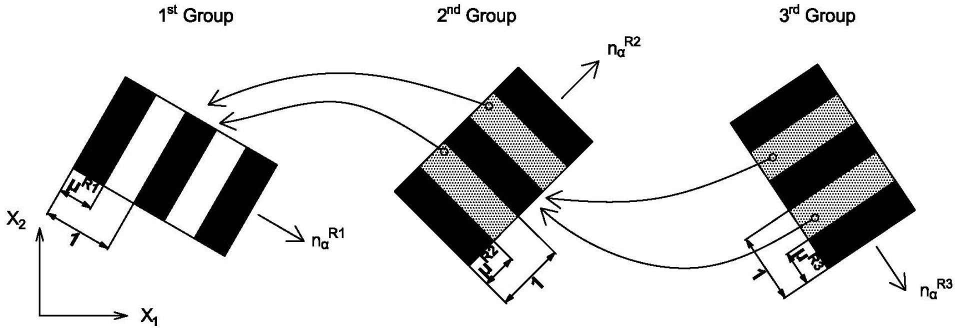


Figure 2.2. Two-dimensional microstructure with 3 groups of layers [Olhoff & Eschenauer, 1999]

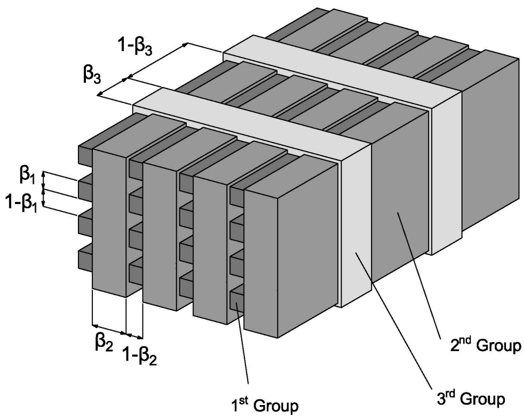


Figure 2.3. Three-dimensional microstructure with 3 groups of layers [Olhoff & Eschenauer, 1999]

2.2.2. Solid Isotropic Material with Penalty (SIMP)

The methods that have been previously commented, are not advisable when the number of elements in that the domain is divided is elevated. This circumstance is consequence of the high number of design variables that have to be used in the definition of the microstructure of each element of the mesh. As a result of this, it was necessary to develop other alternative approaches that mitigate this drawback.

Among all of the methods that have been developed, the most important is the Solid Isotropic Material with Penalty (SIMP). The SIMP method was based on the ideas that had been previously proposed by [Rossow & Taylor, 1973]. The effective elasticity tensor of the material for intermediate values of the density could be calculated in this method as:

$$E_{ijkl} = \rho^p E^0_{ijkl} \qquad p > 1 \qquad 0 < \rho_{min} \leq \rho \leq 1 \tag{2.1}$$

where E_{ijkl} is the elasticity tensor of the material for intermediate values of ρ and E^0_{ijkl} is the value of this tensor when the cell is completely full of material. Furthermore, the exponent $p > 1$ was used to penalize the existence of intermediate values of ρ. On the other hand, the penalty effect over the stiffness of the material can be observed in figure 2.4.

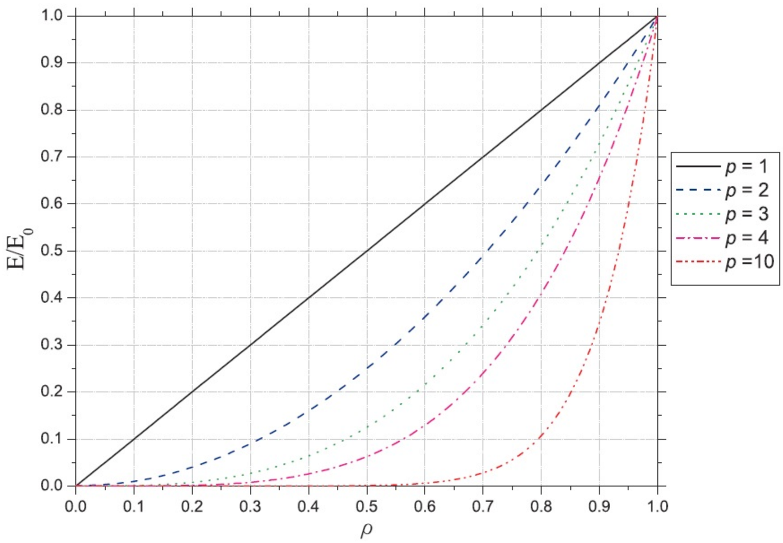


Figure 2.4. Penalty effect over the stiffness in the SIMP method

Even though the introduction of the penalty coefficient could be an arduous task, the effect of this new parameter ended up being advantageous, because more full-void solutions were obtained as the value of this coefficient was increased. This phenomenon was demonstrated by Bendsøe in [Bendsøe, 1995] and [Bendsøe & Sigmund, 1999].

The extensive use of the SIMP method was based on its simplicity, because only one design variable per element had to be used in the definition of the material layout, and it was also possible to attain full-void solutions. On the other hand, the SIMP method was not originally established in the same way as the previous methods. The previous methods taking physical aspects into consideration by means of the definition of a microstructure, nevertheless, the SIMP method was initially proposed from a theoretical point of view. And, once the SIMP model was validated in practice, it was demonstrated that there was at least one real microstructure that represents faithfully the behavior of the theoretical formulation that had been developed beforehand.

The physical interpretation of the SIMP approach was developed in [Bendsøe & Sigmund, 1999]. Finally, an example of the microstructure that had been obtained can be observed in figures 2.5 and 2.6.

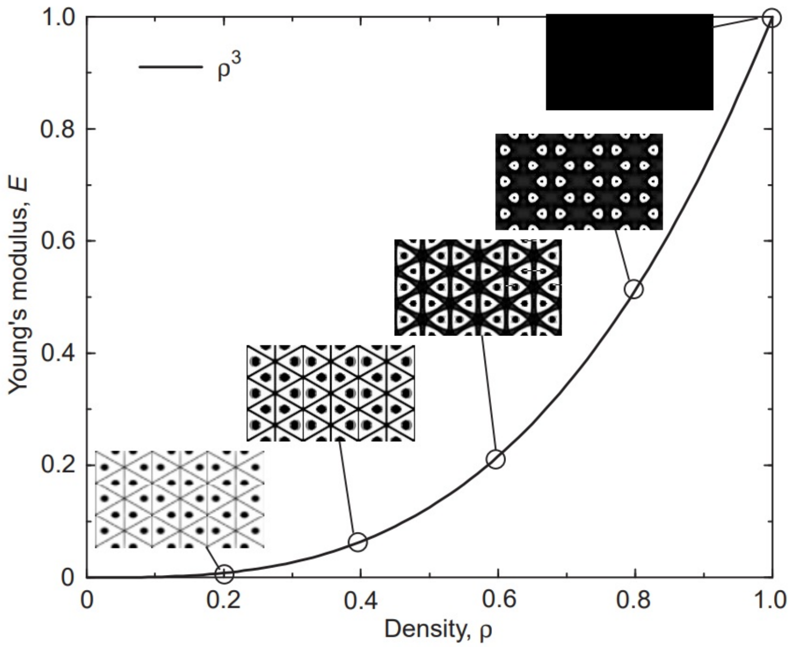


Figure 2.5. Microstructures of material and void realizing the material properties of the SIMP model with $p = 3$, for a base material with Poisson's ratio $\nu = 1/3$. [Bendsøe & Sigmund, 1999] [Eschenauer & Olhoff, 2001]

In spite of all the advantages and improvements that were provided by the SIMP approach, it was also important to take into account all the disadvantages of this method. Thus, the solutions obtained depend on the penalty coefficient. Furthermore, it was also possible to obtain checkerboard solutions and as it was expected, the mesh dependency effect also appeared in the solutions. All this circumstances had been analyzed in [Bendsøe, 1995].

2.2.3. Multimicrostructural approach

Although the previous methods had been widely used, they have an important limitation. In the first formulations of this kind of methods, the microstructure of all the elements in that the domain is divided had to be chosen, beforehand. This option could be reasonable for maximum stiffness problem where the global stiffness is evaluated. It could not be the better strategy for minimum weight with stress constraints problem, since the isostatic lines does not have the same orientation in all the domain.

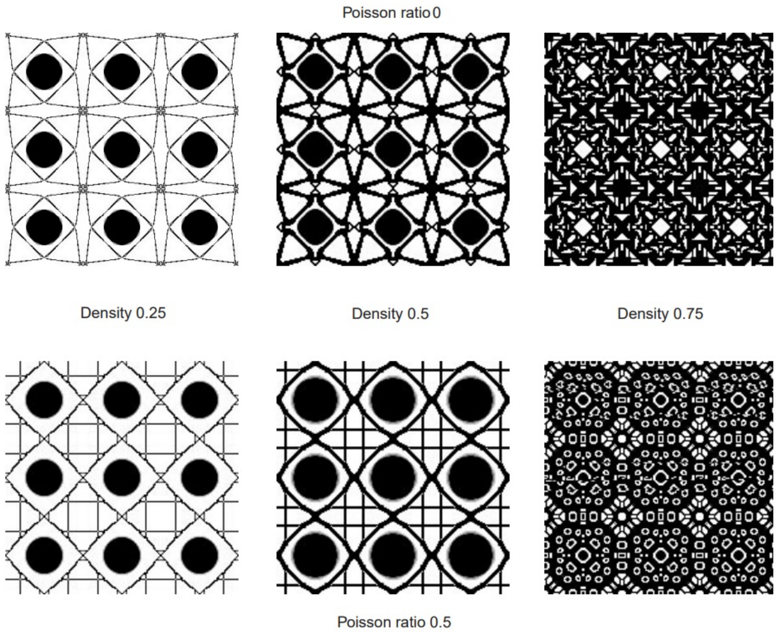


Figure 2.6. Microstructures of material and void realizing the material properties of the SIMP model with $p = 4$, for a base material with Poisson's ratio $\nu = 0$ and $\nu = 0.5$, respectively. [Bendsøe & Sigmund, 1999]

As a result of this, the use of different kinds of microstructure in the same problem as a way to solve the topology optimization problem was developed, because it can be possible that the amount of material that have to be used diminished if the microstructure can be changed. Even though this method was firstly developed by [Rodrigues et al., 2002], since then, this approach has been used extensively. An example where all the microstructures that have to be used in order to obtain the optimal solution of a minimum weight with stress constraint problem can be seen in figure 2.7.

Despite of the advantages that suppose the possibility of changing the microstructure, this possibility introduces new drawbacks. When the number of elements in that the domain is divided is extremely high, the solution of the problem can be irresolvable. In this case, the design variables are not only the amount of material in each part of the domain, but also the microstructure that minimizes this amount of material. Finally, this problem is increased as the number of possible microstructures goes up.

As a consequence of this, there are several strategies to reduce this drawback. One alternative can be to divide the domain in a certain number of regions and to choose the same microstructure for all the elements of each region. This was an approach that has been developed in [Sivapuram et al., 2016], [Xia & Breitkopf, 2014] and [Nakshatrala et al., 2013], among others.

In other alternatives, instead of dividing the domain in a certain number of regions, the microstructures that have to be used, are directly related to the amount of material

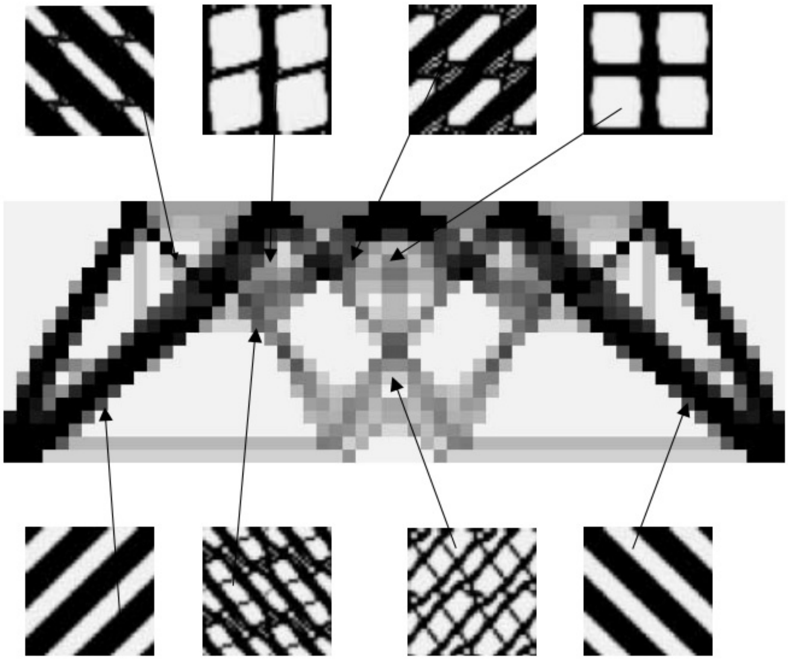


Figure 2.7. Set of microstructures for a stress based solution [Rodrigues et al., 2002]

that have to be used in each element, what supposes an important complexity reduction of the problem. At this point, it is important to take into consideration the number of microstructures that will be used, because the quality of the solution improves as the number of microstructures is increased. In other words, the number of intervals in that the design variable values are divided has to be increased. This approach has been developed in [Zhang et al., 2018] and [Gao et al., 2019], among others.

2.3. Structural weight definition: continuous formulations

Although the discontinuous formulations of density, where the domain is divided in a certain number of elements and the density in each element is independent of the rest, are being extensively used nowadays, they do not provide with a unique optimal solution because of the mesh dependency. The mesh dependency phenomena are concomitant of all the formulations in that the domain has to be meshed.

Although the incorporation of a continuous formulation of the density in these approaches will not avoid the mesh dependency appearance in the solution, it will diminish its effect. This circumstance makes necessary the development of an alternative formulation that does not divide the domain in a certain number of elements in order to avoid the mesh dependency phenomenon. As a result of this, a continuous formulation for the density at all the points of the domain has been established.

Nevertheless, the use of continuous formulations of the density has their own drawbacks, since the direct relationship between the element and the design variable disappears. This is due to the fact that the domain does not have to be divided in elements or the design variables are assigned to points instead of elements. On the other hand, different ways to define the density with a continuous behavior has been established. Even though a deep analysis of all these methods will be developed. A brief explanation of them will be commented next.

There are methods where the value of the density can be calculated multiplying the density in a certain set of control points times the value of their shape functions, like in the Isogeometric Analysis. Other methods calculate the value of the density through the use of an auxiliary function that will be calibrated during the optimization procedure, like the Level Set Method or the Phase Field Approach.

Finally, there are also methods in that the procedure is divided in two phases, like the Bubble Method. In the first one, the part of the domain where there must be material is determined, and in the second one, the amount of material that have to be used is established. Lastly, the different ways density can be defined in a continuous way will be briefly described in the next sections.

2.3.1. Level Set Method

Although the Level Set Method was the first method that did not use the concept of design variables as a way to define the material layout in the domain, it took several years between its development and its application in the topology optimization field. The first work that has been developed in this area was proposed by [Sethian & Wiegmann, 2000]. Since then a big amount of contributions in this approach has been made in order to improve their operation. Among all these contributions the most important were developed in [Wang et al., 2003], [Park & Youn, 2005], [Amstutz & Andrä, 2006] and [Liu & Korvink, 2008].

In the Level Set method, the way material layout is defined consists on establishing an auxiliary function whose value determines the presence or absence of material. Moreover, the value of this function will also determine the amount of material that have to be placed in each point of the domain. Even though, there are two alternatives to define the level set function. Both are equivalent in practice because the only difference between them is the part of the numerical system that defines the presence of material, negative numbers or positive ones. One of this possible formulations is:

$$\begin{cases} \phi(x) > 0, & x \in \Omega_s \\ \phi(x) = 0, & x \in \Gamma_s \\ \phi(x) < 0, & x \notin \Omega_s \end{cases} \tag{2.2}$$

where Ω_s is the domain of the structure and Γ_s is its boundary. Besides, it is also important to establish what is the function that will determine the amount of material depending on the value of the Level Set Function. In general, the function that has been more commonly chosen is the Heaviside Function $H(\phi)$, whose formulation is:

$$H(\phi) = \begin{cases} 1, & \phi \geq 0 \\ 0, & \phi < 0 \end{cases} \tag{2.3}$$

On the other hand, the other aspect that have to be established is the evolution of the Level Set Function (LSF) in the optimization process. This evolution is generally governed by the Hamilton-Jacobi (HJ) equation like in [Amstutz & Andrä, 2006] and [Liu & Korvink, 2008]. This partial equation in the topology optimization field describes the motion of the material interface due to a design velocity field v. Furthermore, the first-order Hamilton-Jacobi equation can be defined as:

$$\frac{\partial \phi}{\partial t} + \nabla \phi \cdot v = 0 \tag{2.4}$$

where t denotes a pseudo-time that in this case represents the iterations of the optimization process. Finally, a review of all concepts and improvements that had been established with respect to the Level Set Methods since their first developments in the topology optimization field can be found in [van Dijk et al., 2013].

Lastly, since this review of the level set method, some of the most important improvements and developments had been: the incorporation of Isogeometric Analysis concepts in the level set method by [Shojaee et al., 2012], [Wang & Benson, 2016] and [Roodsarabi et al., 2016]; the introduction of an automatic hole insertion mechanism by [Dunning & Kim, 2013]; the consideration of the stress constraints in the topology optimization problem by [Jr. & Fancello, 2014] or the incorporation of the topological derivative concept in the level set method by [Baiges et al., 2019].

2.3.2. Bubble Method

The second method that has been incorporated in the topology optimization field as a way to define the material layout is the Bubble Method. This method that has been developed by [Eschenauer & Schumacher, 1993] is a conceptually different method than Level Set. The most important difference is the operation of the method that consist on two steps. In the first one the optimal layout of the material is established depending on the variation of the boundary geometry and in the second one, once the contour of the structure is determined, the introduction of an infinitesimal hole in the most appropriated point of the domain (that is known as ”bubble”) is considered.

When the hole is incorporated, the structure has one topology order more and the general procedure will have to be repeated again until an optimal material layout is achieved. If the infinitesimal hole is inside the material region, the topology of the structure will be modified. However, if the hole is in the border of the structural domain, the structural topology is not modified and the mesh of this area has to be refined in order to obtain a better solution of the problem.

Finally, the procedure finishes when the maximum number of holes that have to be incorporated is achieved or if the size of the holes that are being introduced is lower to a certain established value. Due to the fact that this method does not have a characteristic equation, a graphic representation of the procedure can be observed in figure 2.8.

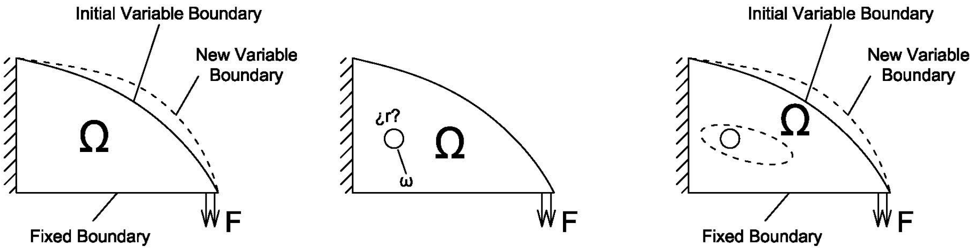


Figure 2.8. Steps of the Bubble Method [Eschenauer et al., 1994]

On the other hand, although this method has the advantage of provide full-void solutions and make the hole incorporation possible, the computational requirements are extremely elevated, especially with respect to the mesh redefinition in some iterations. Therefore, despite of this advantage, this method is not being widely employed nowadays, because of this drawback. Moreover, the most important advantage of the Bubble Method: the hole insertion technique, has been recently developed for other methods, like the Level Set Method in [Dunning & Kim, 2013]. Finally, other reason that explains the reduced use of the Bubble Method, is the complexity of its formulation, that is few intuitive.

2.3.3. Phase Field Approach

Other alternative method that has been used in order to define the material layout in a continuous way is the Phase Field Approach. Although initially this method was not properly developed for topology optimization problems, its similarity with respect to the approach where was being applied, made its application possible in the topology optimization field. This circumstance was due to the fact that the Phase Field Approach was being applied in problems with at least two different phases. And in the topology optimization problem, there are two phases: solid and void.

On the other hand, one of the first works that has been developed in the topology optimization field was [Wang & Zhou, 2004]. Since then a big amount of contributions of this approach has been made in order to improve their operation.

First, the introduction of stress constraints in the topology optimization problem with the Phase Field Approach was developed in [Burguer & Stainko, 2006] and [Jeong et al., 2014]. Then, the incorporation of Isogeometric Analysis approach was established in [Dedè et al., 2012]. Finally, a complete analysis of different Phase Field Approaches was carried out in [Blank et al., 2012].

In the phase field approach, the way material layout is defined is through the use of the parameter known as phase field parameter. This parameter takes a unitary value in order to represent the solid region and a zero value for the void. The intermediate values of the phase field parameter, physically, represent the interface between solid and void phase, or in other words, the boundary of the optimum structure.

Furthermore, in the phase field methods it is also important to choose the way interface is represented, since the interface between the solid and void regions can be sharp or diffuse. A graphic representation of this circumstance can be observed in figure 2.9. In addition, for this adapted phase field model, the most important and usual phase transitions theory is the Van der Waals-Cahn-Hilliard theory.

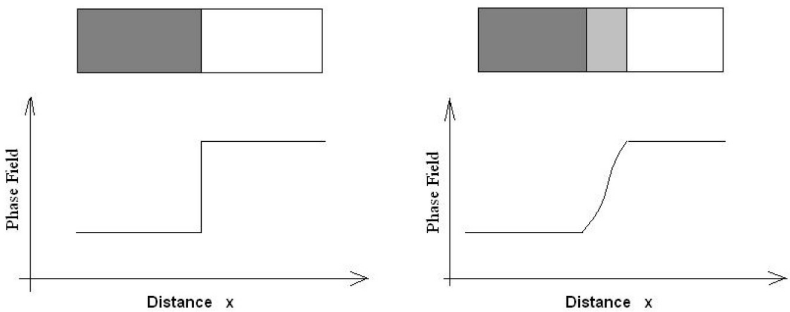


Figure 2.9. The sharp interface (left) and the diffuse interface (right) [Wang & Zhou, 2004]

Last but not least, the structural analysis model has an important influence on the obtained results. For the phase-field approach, let $\Omega \subseteq R^d$ with $(d = 2 \; or \; d = 3)$ is the domain that a linear isotropic elastic structure occupies. The boundary of Ω consist of three parts: $\Gamma = \partial\Omega = \Gamma_0 \cup \Gamma_1 \cup \Gamma_2$, with Dirichlet boundary conditions on Γ_1 and Neumann boundary condition on Γ_2. Apart from that, the boundary Γ_0 is assumed as traction free and the displacement field $\mathbf{u}$ in Ω will be the unique solution of the linear elastic system:

$$\begin{aligned} &-div\,\{\sigma(\mathbf{u})\} = f && \text{in } \Omega \\ &\mathbf{u} = \mathbf{u}_0 && \text{on } \Gamma_1 \\ &\sigma(\mathbf{u}) \cdot n = \mathbf{h} && \text{on } \Gamma_2 \end{aligned} \tag{2.5}$$

where $\mathbf{u}$ is the displacement vector, $\sigma(\mathbf{u})$ is the stress tensor, f the applied body force and $\mathbf{h}$ the boundary traction force. A graphic representation of the problem can be observed in figure 2.10. Even though the results obtained with this method could have been promising, they were not good enough, because of the appearance of a perpendicular contact between the domain border and the optimum structural configuration, as it can be watched in figure 2.11.

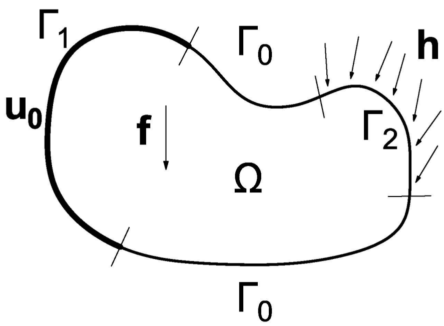


Figure 2.10. Graphic representation of Phase Field problem

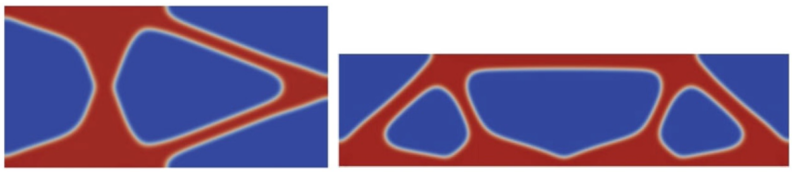


Figure 2.11. Different solutions of Phase Field Approach [Dedè et al., 2012]

The way structural analysis is formulated in terms of the boundary conditions, is the reason for explaining this phenomenon. Due to this, other alternative methods like Level Set are more commonly used nowadays instead of Phase Field Approaches.

2.3.4. Isogeometric Analysis

Finally, the last method that will be described in this thesis and that can be used to establish the density in a continuous way is the Isogeometric Analysis. Although the isogeometric analysis theory was established several years ago, its use as a way to define the material layout in the domain of a topology optimization problem, has been developed recently in [Hassani et al., 2012]. However, the basis of the isogeometric analysis that makes possible their use as an alternative to the Finite Element Method was developed beforehand in [Hughes et al., 2005] and [Hughes et al., 2009].

Unlike the finite element method, in this formulation the value of the density in each point of the domain can be obtained by multiplying the value of the density in each control point times the value of its B-spline shape function as:

$$\rho_{(}\xi, \eta, \zeta) = \sum_{i=1}^{N_x} \sum_{j=1}^{N_y} \sum_{k=1}^{N_z} B_{i,j,k}(\xi, \eta, \zeta) \rho_{i,j,k} \tag{2.6}$$

Despite the fact that this method had already been used as a part of the Level Set Method or the Phase Field Approach. The appearance of a perpendicular contact between the domain border and the optimum structural configuration due to the formulation of the boundary conditions with the second one, did not arouse the interest that it has nowadays. This recent interest is due to the fact that the Isogeometric Analysis can be used as an alternative to the Finite Element Method. This circumstance has been tested in other fields apart from the topology optimization one. Finally, the Isogeometric Analysis as a way to formulate the material layout has been used among others in [Hassani et al., 2012], [Qian, 2013], [Wang et al., 2017], [Lieu & Lee, 2017] and [Liu et al., 2018].

2.4. Stress constraints

Although, initially, the topology optimization problem was stated in terms of maximum stiffness. This situation was due to the fact that the formulation of this problem was very simple, since only the structural global stiffness has to be calculated. Moreover, the solution of this kind of problems does not need an important amount of computational requirements. However, its usefulness has been limited from the engineering point of view, since it is possible that the solutions obtained do not resist all the applied loads, as there are no stresses verification.

Despite these circumstances, the maximum stiffness approach continues being the most studied approach in the topology optimization field and all the possible improvements are firstly tested with it, instead of doing this verification directly with the stress constraints approach. At this point, it is important to introduce the first publications about the minimum weight with stress constraints approach, that were developed in [Duysinx & Bendsøe, 1998], [Muiños et al., 2002], [Navarrina et al., 2002] and [Navarrina et al., 2003].

On the other hand, when the stress constraints were initially formulated, an extensive analysis of them was made in order to solve all the possible drawbacks. Moreover, the stress criterion that had to be used to control the value of the stresses had to be chosen according to the material that is being used in the definition of the problem.

From the beginning, the stress constraints approach aroused a great interest, since it was possible to design the lightest structure that resists all the applied loads. This possibility makes that this approach has a high usefulness in the engineering scope of application. However, the solution of the minimum weight with stress constraints problem became too expensive as the number of stress constraints was increased. Furthermore, it is important to take into account the relationship between the number of design variables, if they exist, and the number of stresses that have to be considered. Finally, if the number of stress constraints is not sufficient, the solution that will be obtained could not be the optimal solution, since the design variables that define the relative density in the parts of the domain without stress constraints will tend to be zero.

Moreover, once the minimum number of stresses that have to be tested was established, it was concluded that the three-dimensional problems can not be solved by the stress constraints approach directly, since the computational resources that have to be used in the solution were extremely high, or in other words, the amount of time that would have to be used to solve these problems would not be competitive for the designers and the industry.

2.4.1. Stress constraints aggregation

Topology optimization of two-dimensional structures can be solved in an acceptable amount of time. However, the need to solve three-dimensional problems made necessary the development of new strategies to check the stresses. These strategies will have to be cheaper with respect to the computational requirements and CPU time in comparison to the previous one. Moreover, new models proposed will have to be tested in two-dimensional problems in order to guarantee that the CPU time in three-dimensional models would be affordable. In other words, if the amount of time used in the solution of two-dimensional problems is reduced, the same will happen with the three-dimensional ones.

On the other hand, the new stress constraints have been established by means of formulas that let estimate the maximum in a certain group of values. In this case the maximum stress in a certain number of calculated stresses. This kind of strategies are known as Stress Constraint Aggregation, and were developed in [Martins & Poon, 2005], [Poon & Martins, 2007], [París et al., 2007a], [París et al., 2007b], [París et al., 2007c], [París et al., 2008] and [París et al., 2010].

However, when the stress constraints aggregation is used, it is not possible to ensure that all the stresses that are being tested do not exceed their limit maximum. Despite of this, it is an assumable cost that has to be accepted if a reduction in the amount of time employed in the solution of the topology problem is intended.

Furthermore, there are different strategies that can be chosen according to the precision that is intended when the topology optimization problem is solved. They are: the global aggregation approach, if all the stresses are aggregated in only one constraint, and the block aggregation approach if the stresses are grouped in a certain number of constraints. The results obtained with the aggregation techniques will be improved as the number of blocks is increased. Nonetheless, the CPU time required by the problem will be increased with this improvement, since the maximum number of blocks will be the number of stresses that are being analyzed. In other words, the extreme cases of the block aggregation are the global (just 1 block) and the local approaches (1 block per local stress constraint) respectively.

2.5. Stress constraints aggregation techniques

Although the way stresses are aggregated can vary with respect to the number of constraints that will be defined, all the formulas that will be introduced in this section can be used for all this strategies of aggregation. Despite the fact that there had been different approaches with respect to the formulation of the stresses aggregation, all the formulas that will be commented in this chapter had been analyzed in [Lambe et al., 2017]. First of all, the stress criterion that have to be introduced in the aggregation formulas will be:

$$g_i(\sigma_i) = \frac{\sigma_{v,i}}{\sigma_{d,i}} \tag{2.7}$$

where i is the point where stress is computed, σ_v is the von Mises stress and σ_d is the maximum allowable stress. The stress constraint for the aggregation approach can be written as:

$$\max\{g_i\} \leq 1 \tag{2.8}$$

Even though, this constraint is equivalent to all the stress constraints that have to be considered in the formulation of the problem, since all stresses have to be lower to their maximum allowable value. The main problem is, however, to ensure that the maximum value obtained is at least equal to the real maximum value of all the aggregated terms. For this reason, four different strategies to calculate this maximum will be presented. Two of them are more classical ([Kreisselmeier & Steinhauser, 1979] and [Duysinx & Sigmund, 1998]) and the other two had been developed more recently ([Kennedy & Hicken, 2015]).

2.5.1. KS aggregation

The first classical strategy that has been used to aggregate whatever constraint is the Kreisselmeier-Steinhauser (KS) function. This approach was first developed in [Kreisselmeier & Steinhauser, 1979] and [Kreisselmeier & Steinhauser, 1983].

Although there is a formula for the aggregation of continuous constraints in the domain of the problem. The formula that had been used in case of discrete constraints, such as the stress constraints, is the discrete constraints aggregation formula that is:

$$C_{DKS}(\rho) = \frac{1}{\rho} \ln \left[\sum_{i=1}^{N_c} e^{\rho g_i} \right] \tag{2.9}$$

where ρ is the aggregation parameter. As it can be watched, in this method an exponential combination of all values is done. On the other hand, in the topology optimization field, the KS aggregation formula was introduced as a way to aggregate the stress constraints in [París, 2007], among others.

2.5.2. p-norm aggregation

The second classical technique whereby the constraints can be aggregated is the p-norm approach. Although this method was firstly developed in [Duysinx & Sigmund, 1998], it was improved later in [Qiu & Liu, 2010].

The p-norm Aggregation have some similarities with the KS aggregation. First, there is a general equation for continuous constraints. Moreover, there is an equation for discrete constraints like the stress constraints. Therefore, the equation that have to be used in this case is:

$$C_{DPN}(\rho) = \left[\sum_{i=1}^{N_c} |g_i|^{\rho} \right]^{\frac{1}{\rho}} \tag{2.10}$$

where ρ is the p-norm of this group of constraints. In this case a power combination of all values is done. On the other hand, it is also important to take into consideration the fact that only positive or negative values can be combined directly. The method does not work properly if negative and positive values are combined. Finally, this aggregation approach was also analyzed and considered in [París, 2007], among others.

2.5.3. Induced constraint aggregation

Apart from this two classical aggregation constraints approaches, recently, two new approaches have been formulated. Both methods have been developed in [Kennedy & Hicken, 2015]. Moreover, they are equivalent to KS aggregation and p-norm aggregation in a certain sense. These approaches are known as induced aggregation methods. Furthermore, they try to obtain a better accurate approximation when the maximum value of a certain group is estimated for whatever value of ρ.

On the other hand, the demonstration that these approaches are equivalent to the previous one, when the parameter ρ tends to infinity has been developed. In the same way that for the classical approaches there are a formula for continuous constraints. However, in the topology optimization field, the most interesting formula is the discrete. The formulas of both methods for discrete constraints are:

$$C_{DIE}(\rho) = \frac{\sum\limits_{i=1}^{N_c} g_i e^{\rho g_i}}{\sum\limits_{i=1}^{N_c} e^{\rho g_i}} \tag{2.11}$$

$$C_{DIP}(\rho) = \frac{\sum\limits_{i=1}^{N_c} g_i^{\rho+1}}{\sum\limits_{i=1}^{N_c} g_i^{\rho}} \tag{2.12}$$

where C_{DIE} and C_{DIP} are the induced exponential functional and the induced power functional, respectively. Similarly, to the p-norm, the induced power functional is only suited for strictly positive or negative functions g. In other case, the aggregation is not effective.

Finally, one advantage of these induced aggregation functions over the classical aggregation function is the convergence to a single value for an arbitrary number of sample points. This is evident since the induced functions are ratios of sums rather than single sums.

2.6. Multiresolution techniques

Although the main objective of the topology optimization has always been the attainment of the optimal solutions subjected to certain conditions. The attainment of solutions with a high spatial definition has been a secondary aim once the topology optimization problem has been solved. However, this objective requires the implementation of an extremely accurate structural analysis and the use of an important number of design variables. This objective is hardly achievable now due to the extremely high computational requirements.

As it can be seen in the previous section, the computational requirements needed to solve the topology optimization problem have been reduced by using constraints aggregation techniques. Nevertheless, when these techniques were introduced, the reduction of time only affected the optimization process. As a result of this, the part that required most of the CPU time was the structural analysis, especially when the number of elements was increased.

As a consequence of this, a new strategy to calculate the structural analysis had to be developed, in order to reduce, among other things, the CPU time without losing accuracy. These new approaches are known as Multiresolution Technique and it had been initially developed in [Kim & Yoon, 2000] and [Nguyen et al., 2010].

On the other hand, the Multiresolution Techniques consist on using different schemes of discretization for the structural analysis and the material layout. In other words, the number of elements in that the mesh is divided in order to calculate the structural analysis will be different to the number of design variables that will be used in the definition of the material layout.

Furthermore, the number of elements that will be used in the structural analysis mesh will be generally lower than the number of design variables. That makes possible the attainment of solutions with a high spatial definition and a precise calculation of the structural analysis with a lower amount of computational requirements. A scheme of the meshing procedure can be observed in figure 2.12 and in figure 2.13.

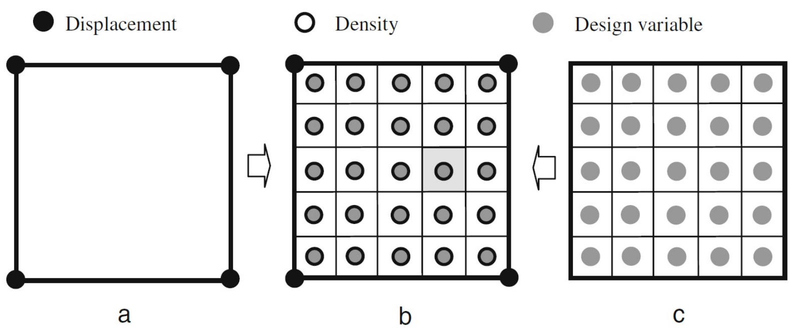


Figure 2.12. MTOP Q4/n25 element: a) displacement mesh, b) superposed meshes, c) design variable mesh [Nguyen et al., 2010]

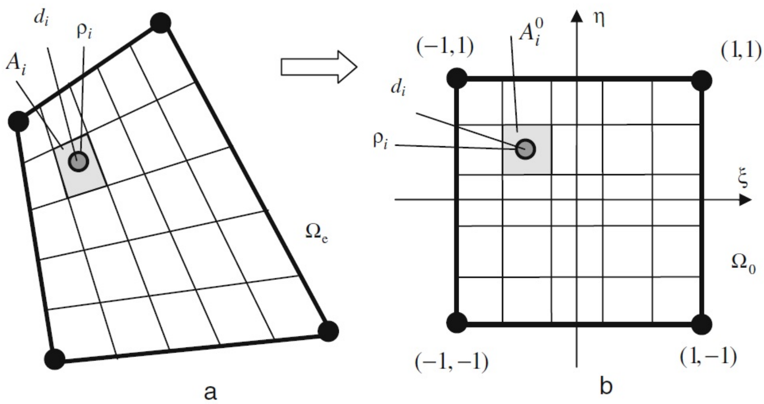


Figure 2.13. Isoparametric element: a) initial, b) parent domain [Nguyen et al., 2010]

Once this method had been established, some improvements of them were introduced in [Nguyen et al., 2012]. The most important difference that supposes this improvement is the way material layout is defined. Instead of using only one mesh, two meshes are employed. One of this meshes is used for the definition of the material layout by means of a density value, and the other mesh for the establishment of the design variables whose value will define the density in each element. A graphic scheme of this new approach can be watched in figure 2.14.

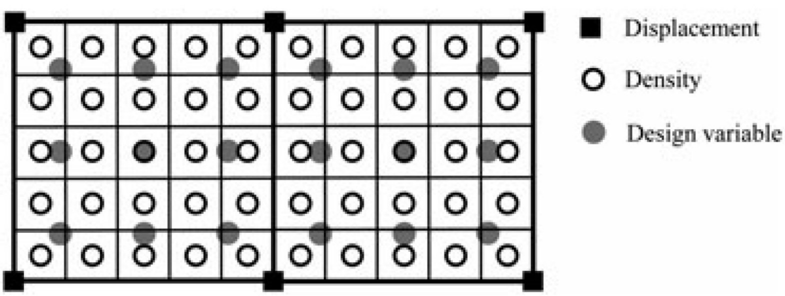


Figure 2.14. Displacement, density and design variables meshes [Nguyen et al., 2012]

Finally, this method has been also implemented in three-dimensional topology optimization, not only in [Nguyen et al., 2010], but also in [Park & Sutradhar, 2015]. The procedure whereby the displacements, densities and design variables are defined is the same that for two-dimensional cases.

2.7. Topological derivative

In the same way as the optimization problems the concept of shape and topological derivative was firstly introduced in the shape optimization problems [Sokolowski & Zochowski, 1999] and [Schumacher, 1995] and later in the topology optimization ones [Burguer et al., 2004] and [Novotny et al., 2007]. Both concepts, shape derivative and topological derivative, are quite similar since the former is based in local perturbations of the boundary of the structural domain and the latter measures the influence of small holes introduced (topology perturbations) in the structural domain.

The topological derivative compares one of the characteristics of the structure such as the structural stiffness or the structural stresses of two models: the structure without hole and the same structure with an infinitesimal hole. The main objective is to analyze the influence that the introduction of this hole has in the characteristic chosen.

Since the main objective of this thesis is to solve a topology optimization problem of minimum weight with stress constraints, the characteristic chosen will be hereinafter the structural stresses. Therefore, the main objective of the topological derivative will be to analyze the parts of the structural domain where a removal of material has less influence over the stresses that are close to their maximum allowable value.

For this purpose, the use of the stress constraints aggregation techniques would be recommended since it will be only necessary to analyze the influence that the hole insertion has in the value of this constraint. Once the topological derivative has been computed, the material places in the parts of the domain with the lowest influence in the value of the global constraint is removed.

Then, the process is repeated iteratively until the hole insertion only produces a violation of the global constraint, in other words, it is not possible to remove more material without overcoming the maximum allowable stresses. Finally, this method has been also developed with different approaches in [Giusti et al., 2008], [Novotny et al., 2015], [Ferrer, 2017], [Baiges et al., 2019] and [Martínez-Frutos et al., 2019].

2.8. Damage approach

In the previous sections a review about similar methods in that stress constraints are imposed directly in the topology optimization problem by means of a comparison between the stress value and their maximum allowable value, and the use of the constraints aggregation techniques has been presented. However, there are other alternative approaches for the imposition of the stress constraints in an indirect way.

The main advantage of these indirect techniques is a smaller CPU time. However, the previous constraints aggregation techniques have an inherent handicap, since it is not possible to ensure that all stresses will be lower than their maximum allowable value. As a consequence of this, a new indirect approach that is known as a Damage Approach had been developed in [Verbart et al., 2016].

In the Damage Approach, an alternative model of the element that is being designed is defined in accordance with the relationship between the stress in a certain set of points and their maximum allowable value. Moreover, if the stresses are higher than their maximum allowable value, the alternative model will be penalized or, in other words, damaged.

On the other hand, it is important to choose one general characteristic of the structure that will be penalized. In [Verbart et al., 2016], the property that had been chosen is the structural stiffness. As a consequence of the penalization the alternative model will be less rigid when any stress exceeds its maximum allowable value. Nonetheless, structural stiffness can be replaced by other properties, but these approaches have not been analyzed until now.

Finally, the way constraint was defined is by means of a comparison of this structural characteristic between the original model and the alternative one. This way to establish the constraints allows to ensure that all stresses will be lower than their maximum allowable value, what means an important improvement with respect to the majority of the constraints aggregation techniques.

2.9. Overview

Since the basis of the topology optimization problem were established by Bendsøe and Kikuchi in 1988 [Bendsøe & Kikuchi, 1988], a big amount of contributions in this field was made, due to their wide scope of application. Although two different kind of problems had been mainly established, the maximum stiffness problem and the minimum weight with stress constraints one, in this thesis only the second problem will be analyzed.

In the first instance, the different discontinuous ways to define the material layout were analyzed. In these approaches, the domain is always divided in elements, and there is no continuity between the amount of material used in adjacent elements. However, the mesh dependency due to the division of the domain made impossible to obtain a unique optimal solution, since if the mesh is modified the solution obtained changes. On the other hand, the microstructure of each element of the mesh can be defined by means of a microstructure catalog. Therefore, if the microstructure is changed in a certain element, it is possible that the solution obtained will be also modified. This way to define the material layout will be used with the Finite Element Method.

In the second instance, the continuous formulations of the material layout were established. Although it is possible to avoid the mesh dependency phenomenon, the way material layout was defined in these approaches is more complicated due to the no existence of design variables. However, the isogeometric analysis in that the mesh dependency is also present makes possible to diminish the influence of this phenomenon by means of the use of more design variables for the same number of elements in comparison to the discontinuous formulations.

On the other hand, formulations like the Phase-Field Model give solutions that shows a particular characteristic. The reason whereby the results are not adequate it is unknown at this moment. Moreover, there are methods that are not used nowadays, as the Bubble Method, since the Level Set Method gives better features and performances.

Despite the mesh dependency, in this thesis the isogeometric analysis will be implemented as an alternative to the Finite Element Method not only in the definition of the Material Layout but also in the Structural Analysis.

Apart from that, the different strategies that can be used to impose stress constraints has been analyzed. However, a new approach will be used in this thesis to introduce the stress constraints in the topology optimization problems, because of the main drawbacks of the previous methods.

At this point, it is important to take into consideration the conflict between the CPU time required and the assurance that all stresses will be lower than their maximum allowable value. This circumstance is only sure in the local approach, when each stress has its own constraints. Furthermore, the most typical formulations used to aggregate stress constraints had been also commented.

Additionally, the concept of multiresolution techniques has been also introduced. In this approach, the main objective is to obtain accurate results in a little amount of time. And for this reason different meshes have to be used in the definition of the problem. Despite of it, these techniques will not be used in this thesis, because of the complexity that works with two different meshes can introduce in the formulation of the problem.

Then, the concept of topological derivative has been introduced. In this approach, the main objective is to measure the influence that the introduction of the small holes in the structural domain has over the objective function or the constraints of the topology optimization problem to establish the parts of the domain where is more advisable to remove material.

Finally, the Damage Approach has been briefly defined. This approach to impose stress constraints will be used in this thesis, considering the structural weight instead of the structural stiffness as a general characteristic to be penalized.

Once the topology optimization of structures has been completely introduced and the state of the art in this field has been analyzed, the next step is the development of the continuous formulation of the minimum weight with stress constraints approach that will be proposed in this thesis. Furthermore, in the next chapter, the main concepts of the isogeometric analysis will be introduced. Isogeometric analysis will be used both for the definition of the material layout and for the structural analysis.

Chapter 3

Isogeometric Analysis

"That's one small step for man, one giant leap for mankind"
Neil Armstrong, (1930-2012).

3.1. Introduction

Isogeometric analysis is one of the methods that will be proposed in this thesis in order to do the structural analysis and the material layout in the design domain. The reason whereby this approach will have their own chapter, is its reduced application in the topology optimization field, in comparison to the Finite Element Method, whose use is more extensive.

However, Isogeometric Analysis and Finite Element Method have a lot of features in common, since they are based on the same criteria: Smooth Shapes and the Mean Weighted Residuals (MWR). Despite of this, Isogeometric Analysis is more geometrically based than Finite Element Method, since it takes inspiration from Computer Aided Design (CAD) and its main scope of application has been related with the design step of elements instead of the calculation step of them.

The Finite Element Method was used for the first time in the 1950s-1960s. This approach was firstly used in the civil engineering field, in the calculation of structures and buildings. It was not until the late 1960s when the first commercial computer programs appeared. Because of its earlier development, the Finite Element Method spread to other engineering and scientific disciplines, and nowadays its use is widespread and there are many commercial programs available in spite of the disadvantages with respect to the Isogeometric Analysis.

Meanwhile, Computer Aided Design, what is the basis of the Isogeometric Analysis, had its origins in the 1970s-1980s, that supposes a later development in comparison to the Finite Element Method. Despite the fact that geometry is the underpinning of analysis. Finally, an introductory book, with historical insights was written by Rogers [Rogers, 2001].

This difference at the time when both methods were developed explains why the geometric representation in finite element methods and CAD is so different nowadays. The majority of finite element programs were technically mature long before modern CAD programs were widely adopted.

On the other hand, the use of CAD programs was not only limited to make analyses, it could be also used to design complex geometries, for this reason, it is widely employed in automobile, aeronautical and aerospace engineering, and all this is possible due to its origin.

In this thesis, B-splines will be used to represent the material's density and also the structural behavior instead of NURBS (Non-Uniform Rational B-Splines) [Liu et al., 2018]. The representation could be made by means of surfaces or volumes depending on the structural problem. In spite of not being a requisite in isogeometric analysis, NURBS are the most thoroughly developed CAD technology. This circumstance makes NURBS are the one with most widespread use. Moreover, both NURBS and B-splines allow the attainment of better results about continuity and derivability regarding to the Finite Element Models. However, the use of B-splines is computationally cheaper than the use of NURBS.

The main disadvantages of Finite Element Methods in comparison with Isogeometric Analysis are related by one hand with mesh generation; what is an expensive task, sometimes affected by errors. On the other hand, they are related with interpolation models, because geometry is modified when the mesh is refined and only C^0 continuity is guaranteed. By the contrary in typical CAD interpolation models, any arbitrarily high order of continuity can be guaranteed and geometry can be kept unmodified as mesh is refined.

The main advantage of Finite Element Method is the high versatility in the meshing process in comparison with Isogeometric Analysis. Moreover, other important difference between both methods is in the interpretation of the results. The values obtained in mesh points coincide with the value in this point in FEM; however, in isogeometric analysis, the values that are obtained, only represent a value in a control point, that not necessarily has to be the same to the value in this point of the domain, in order to obtain the real solution of the problem, an interpolation with B-splines have to be done.

To conclude, the step of adapting FEM code to IGA code only requires to make few changes in the code. A correct understanding of the differences between both methods is necessary, but the improvements that are obtained are worthwhile.

3.2. Formulation

Once the historical review of the Isogeometric Analysis has been developed, and a comparative analysis between the Finite Element Method and the Isogeometric Analysis has been made. It is time to introduce the basic concepts of the Isogeometric Analysis. Although, these concepts will be introduced for one-dimensional cases. Its extrapolation for two-dimensional and three-dimensional problems is not difficult, since it only requires to repeat the same process once or twice respectively. The concepts defined in this section will be: The Knot Vector, the Basis Function, the B-spline curves and the B-spline surfaces and solids.

To begin with, B-splines surfaces and B-splines solids are built from B-splines curves. The B-spline parametric space is local to "patches" rather than elements. In other words, the patches play the same role than the elements in the Finite Element Method. However, the patches in the Isogeometric Analysis, are subdomains within that element types and material models are assumed to be uniform. This circumstance supposes an important difference between the Finite Element Method and the Isogeometric Analysis. In the Finite Element Method, each element has their own shape functions, and in the Isogeometric Analysis, each B-spline surface and B-spline solid define the shape functions in a certain set of elements, because of this uniformity. Finally, these elements are known as Knot spans.

3.2.1. Knot vector

First, the knot vector is the basis to build B-splines. The knot vector is a set of coordinates in the parametric space, area where the B-splines will be defined. This parametric space will be equivalent with the patch defined previously. Both, parametric space of B-spline surfaces or B-spline solids and patch can have the same dimensions, but it is not compulsory. On the other hand, it will be necessary to define one knot vector for each spatial dimension.

The knot vector has to be written as $\mathbf{\Xi} = \{\xi_1, \xi_2, ..., \xi_{n+p+1}\}$, where $\xi_i \in \mathbb{R}$ is the ith knot, i is the knot index, $i \in \{1, 2, ..., n+p+1\}$, p is the polynomial order, and n is the number of basis functions that define the B-spline. Additionally, there are several aspects that should be taken into consideration when knot vector is defined.

Firstly, the polynomial order of the B-spline p has to be defined. When its value is 0, 1, 2, 3, etc. the B-spline will be constant, linear, quadratic, cubic, etc. respectively. In this case, an equivalence is established between words "order" and "degree". Secondly, it is necessary to establish if the continuity degree of the B-spline have to be modified in any part of the sub-domain. For this purpose, it will be necessary to repeat the same knot as times as the continuity degree of the B-spline in this point has to be reduced. In other words, several knots will have the same coordinate in the parametric space. And they are known as a repeated knot.

On the other hand, if all knots are equally-spaced, knot vector can be called as uniform. Otherwise, it is called non-uniform. Moreover, if the first and last knots appear $p+1$ times, knot vector is known as open. This circumstance is important in the Topology Optimization Problem, since the open knot vectors define the B-spline in all the domain placed between the first and the last control point. In other case, it will be necessary to define control points out of the patch. For this reason, in CAD, open knot vectors are standard.

The most important characteristic of basis functions formed from open knot vectors is that they are interpolatory in the borders of the parametric space interval $[\xi_1, \xi_{n+p+1}]$. This means that in case of multiple dimensions, basis functions are only interpolatory in the corners of the domain, but they are never interpolatory at interior knots. As it was mentioned previously, this is a distinguishing characteristic between knots in isogeometric analysis, and nodes in finite element method. Once the knot vector has been completely defined, the next step will be to define the Basis Functions of the B-spline.

3.2.2. Basis functions

B-spline basis functions are defined with the Cox - De Boor recursion formula. This procedure has been established by [Cox, 1972] and [de Boor, 1972], and it starts with the next equation:

$$N_{i,0}(\xi) = \begin{cases} 1 & \text{if} \quad \xi_i \leq \xi < \xi_{i+1} \\ 0 & \text{otherwise} \end{cases} \qquad i = 1, ..., n+p \tag{3.1}$$

For $p \geq 1$ basis functions are defined by means of next equation:

$$\begin{gathered} N_{i,q}(\xi) = \left(\frac{\xi - \xi_i}{\xi_{i+q} - \xi_i}\right) N_{i,q-1}(\xi) + \left(\frac{\xi_{i+q+1} - \xi}{\xi_{i+q+1} - \xi_{i+1}}\right) N_{i+1,q-1}(\xi) \\ q = 1, ..., p \qquad\qquad i = 1, ..., n + (p - q) \end{gathered} \tag{3.2}$$

where q is the degree of the B-spline basis function. Moreover, derivatives with respect to parametric coordinates may be computed by way of standard techniques described by Hughes [Hughes, 2000]. An initial example of the results applying previous equations to a uniform knot vector will be presented in figure 3.1.

On the other hand, B-splines basis functions have the next properties:

- Partition of unity

$$\sum_{i=1}^{n} N_{i,p}(\xi) = 1 \qquad \forall \xi \tag{3.3}$$

- The support of each $N_{i,p}$ is compact and contained in the interval $[\xi_i, \xi_{i+p+1}]$
- No negativity of basis functions. Consequently, all coefficients of a mass matrix computed from a B-spline basis will be greater than, or equal to, zero.

$$N_{i,p}(\xi) \geq 0 \qquad \forall \xi \tag{3.4}$$

- If there are not repeated knots, $N_{i,p}(\xi) \in C^{p-1}$ ($p - 1$ order of continuity)

Once the basis function of the B-spline has been defined, the next step will be to define the B-spline curves.

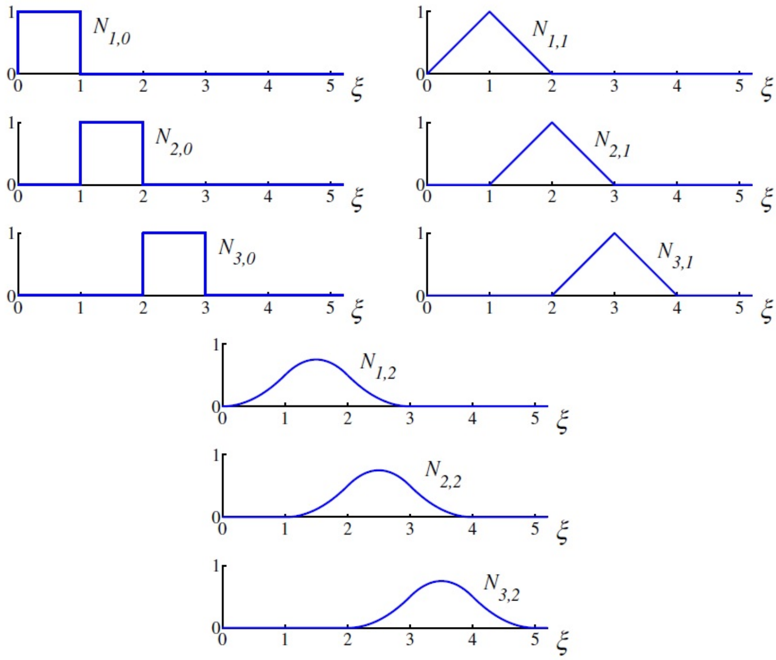


Figure 3.1. Basis functions of order 0,1,2 for uniform knot vector $\Xi = \{0, 1, 2, 3, 4, ...\}$ [Hughes et al., 2005]

3.2.3. B-spline curves

B-spline curves in $\mathbb{R}^d$ are built with a linear combination of B-spline basis functions. Each coefficient of the basis functions is related to a control point. This process is analogous in finite element method, changing control points for mesh nodes.

The control polygon of the B-spline curve is obtained by means of the piecewise linear interpolations of the control points. In other words, this polygon is obtained through the direct connection of all control points in correlative order.

As it was mentioned previously, with the exception of domain's corners that are interpolated by B-spline curves in case of open knot vector, there will not any coincidence between the value in the control points and the value of the B-spline curve in whatever point of the domain.

Moreover, given n basis functions, $N_{i,p}, i = 1, 2, ..., n$, and their corresponding control points $B_i \in \mathbb{R}^d, i = 1, 2, ..., n$, the formula that gives the value of B-spline curve in a certain point is:

$$C(\xi) = \sum_{i=1}^{n} N_{i,p}(\xi) B_i \tag{3.5}$$

On the other hand, B-splines curves have the next properties:

- They have continuous derivatives of order *p-1* in the absence of repeated knots or control points.
- Repeating a knot or control points *k* times diminishes the number of continuous derivatives by *k*.
- Repeating the knot or control point exactly *p* times makes them interpolatory.
- Repeating the knot or control point exactly *p+1* times makes the B-spline curves discontinuous.
- An affine transformation of a B-spline curve can be obtained by means of the application of the transformation to the control points. This property is also known as affine covariance.

3.2.4. B-splines surfaces and solids

In the two-dimensional space, the analogy of B-splines are B-spline surfaces. Given a control net $(B_{i,j}), i = 1, 2, ..., n, j = 1, 2, ..., m$, and knot vectors $\Xi = \{\xi_1, \xi_2, ..., \xi_{n+p+1}\}$, and $H = \{\eta_1, \eta_2, ..., \eta_{m+q+1}\}$, a tensor product B-spline surface is defined by:

$$S(\xi, \eta) = \sum_{i=1}^{n} \sum_{j=1}^{m} N_{i,p}(\xi) M_{j,q}(\eta) B_{i,j} \tag{3.6}$$

where $N_{i,p}$ and $M_{j,q}$ are basis functions of B-spline curves in each dimension. For purposes of numerically integrating arrays constructed from B-splines, "elements" are taken to be knot spans, namely, $[\xi_i, \xi_{i+1}] \times [\eta_j, \eta_{j+1}]$.

In the same way, in the three-dimensional space, tensor product B-spline solids will be generated through B-splines like B-spline surfaces. Given a control net $(B_{i,j,k}), i = 1, 2, ..., n, j = 1, 2, ..., m, k = 1, 2, ..., l$, and knot vector $\Xi = \{\xi_1, \xi_2, ..., \xi_{n+p+1}\}$, $H = \{\eta_1, \eta_2, ..., \eta_{m+q+1}\}$, and $Z = \{\zeta_1, \zeta_2, ..., \zeta_{l+r+1}\}$, a B-spline solid is defined by:

$$S(\xi, \eta, \zeta) = \sum_{i=1}^{n} \sum_{j=1}^{m} \sum_{k=1}^{l} N_{i,p}(\xi) M_{j,q}(\eta) L_{k,r}(\zeta) B_{i,j,k} \tag{3.7}$$

3.3. Implementation

The implementation of isogeometric analysis in a code, in contrast to general belief, does not have any difficulty. The procedure of creating the code is very similar to the typical creation of FEM code.

On the other hand, in the case of having a finite element method code there are several things that have to be considered. First of all, elements in finite element methods are substituted by knot spans in isogeometric analysis. Secondly, nodes are substituted by control points in the same way. Thirdly, shape functions are substituted by B-splines, through Piegl and Tiller algorithms, analogously. Finally, the rest of the code remain unchanged (same weak form, same general organization of the code, same integration formulas, etc.). For all these reasons, the substitution of FEM by IGA is very simple as it was previously mentioned.

The isogeometric analysis has several advantages in comparison to finite element methods:

- CAD models could be analyzed without the need for mesh generation.
- Mesh refinement becomes trivial; the geometry does not have to be modified in this process.
- Continuity can be arbitrarily raised.
- Higher order problems can be addressed.

However, there are several disadvantages, that should be taken into account, when isogeometric analysis is chosen:

- Unknowns no longer represent nodal values.
- Essential boundary conditions may become non-trivial.
- Geometric modelling is less intuitive.

Once all the theoretical aspects of the Isogeometric Analysis that will be used in this thesis have been developed. The next step will be to formulate some of them, since they will be used in the next Chapters. The procedure will be the next: firstly, the knot vector will be defined and then the basis functions will be formulated, finally the variables that will be defined by means B-spline surfaces and B-spline solids will be introduced.

3.3.1. Knot vector definition

In this thesis, a uniform and open knot vector $\boldsymbol{\Xi}$ will be used, but with the objective of obtaining B-splines with 2 orders of continuity, it will be necessary that $p = 2$. For this reason, knot vector in each dimension will have this structure:

$$\boldsymbol{\Xi} = \left\{0, 0, 0, \frac{1}{n}, \frac{2}{n}, ..., \frac{i}{n}, ..., \frac{n-1}{n}, 1, 1, 1\right\} \tag{3.8}$$

where n will have to be n_x, n_y or n_z in each dimension. On the other hand, the number of knots will have to be always $(n+2)+p+1$.

3.3.2. Basis function formulation

B-spline basis functions will be defined with Cox - De Boor recursion formula. In this case, the B-spline basis functions, with $q = 0$ will be:

$$
\begin{aligned}
&N_{1,0}(\xi) = 0\\
&N_{2,0}(\xi) = 0\\
&N_{i+2,0}(\xi) = \begin{cases} 1 & \text{if} \quad \dfrac{i-1}{n} \le \xi \le \dfrac{i}{n} \\ 0 & \text{otherwise} \end{cases}\\
&N_{n+3,0}(\xi) = 0\\
&N_{n+4,0}(\xi) = 0
\end{aligned}
\tag{3.9}
$$

where $i = 1, ..., n$.

In the case that $q = 1$ the B-spline basis functions will be:

$$
\begin{aligned}
&N_{1,1}(\xi) = 0\\
&N_{2,1}(\xi) = \begin{cases} n\left(\dfrac{1}{n} - \xi\right) & \text{if} \quad 0 \le \xi \le \dfrac{1}{n} \\ 0 & \text{otherwise} \end{cases}\\
&N_{i+2,1}(\xi) = \begin{cases} n\left(\xi - \dfrac{i-1}{n}\right) & \text{if} \quad \dfrac{i-1}{n} \le \xi \le \dfrac{i}{n} \\ n\left(\dfrac{i+1}{n} - \xi\right) & \text{if} \quad \dfrac{i}{n} \le \xi \le \dfrac{i+1}{n} \\ 0 & \text{otherwise} \end{cases}\\
&N_{n+2,1}(\xi) = \begin{cases} n\left(\xi - \dfrac{n-1}{n}\right) & \text{if} \quad \dfrac{n-1}{n} \le \xi \le 1 \\ 0 & \text{otherwise} \end{cases}\\
&N_{n+3,1}(\xi) = 0
\end{aligned}
\tag{3.10}
$$

where $i = 1, ..., n-1$.

And finally in case that $q = 2$ the B-spline basis functions will be:

$$N_{1,2}(\xi) = \begin{cases} n^2 \left(\dfrac{1}{n} - \xi\right)^2 & \text{if} \quad 0 \le \xi \le \dfrac{1}{n} \\ 0 & \text{otherwise} \end{cases}$$

$$N_{2,2}(\xi) = \begin{cases} \dfrac{n^2}{2} \left(\dfrac{4\xi}{n} - 3\xi^2\right) & \text{if} \quad 0 \le \xi \le \dfrac{1}{n} \\ \dfrac{n^2}{2} \left(\dfrac{2}{n} - \xi\right)^2 & \text{if} \quad \dfrac{1}{n} \le \xi \le \dfrac{2}{n} \\ 0 & \text{otherwise} \end{cases}$$

$$N_{i+2,2}(\xi) = \begin{cases} \dfrac{n^2}{2} \left(\xi - \dfrac{i-1}{n}\right)^2 & \text{if} \quad \dfrac{i-1}{n} \le \xi \le \dfrac{i}{n} \\ \dfrac{n^2}{2} \left(-2\xi^2 + \dfrac{\xi}{n}(4i+2) - \dfrac{1}{n^2}(2i^2+2i-1)\right) & \text{if} \quad \dfrac{i}{n} \le \xi \le \dfrac{i+1}{n} \\ \dfrac{n^2}{2} \left(\dfrac{i+2}{n} - \xi\right)^2 & \text{if} \quad \dfrac{i+1}{n} \le \xi \le \dfrac{i+2}{n} \\ 0 & \text{otherwise} \end{cases}$$

$$N_{n+1,2}(\xi) = \begin{cases} \dfrac{n^2}{2} \left(\xi - \dfrac{n-2}{n}\right)^2 & \text{if} \quad \dfrac{n-2}{n} \le \xi \le \dfrac{n-1}{n} \\ \dfrac{n^2}{2} \left(\dfrac{4}{n} - 3 + \xi\left(6 - \dfrac{4}{n}\right) - 3\xi^2\right) & \text{if} \quad \dfrac{n-1}{n} \le \xi \le 1 \\ 0 & \text{otherwise} \end{cases}$$

$$N_{n+2,2}(\xi) = \begin{cases} n^2 \left(\xi - \dfrac{n-1}{n}\right)^2 & \text{if} \quad \dfrac{n-1}{n} \le \xi \le 1 \\ 0 & \text{otherwise} \end{cases} \tag{3.11}$$

where $i = 1, ..., n-2$.

The B-spline basis function obtained for $q = 2$ will be used in this thesis to define the density distribution and to calculate structural displacements. An example for $n = 5$ is shown in figure 3.2.

3.3.3. Variables defined by means of B-splines surface and solids

Finally, B-spline surfaces and B-spline solids can be obtained by combining the previous formulas for each dimension and taking into consideration that n have to be replaced by n_x,n_y or n_z respectively. These are the formulas that will have to be used to define density and displacements.

In the two-dimensional space:

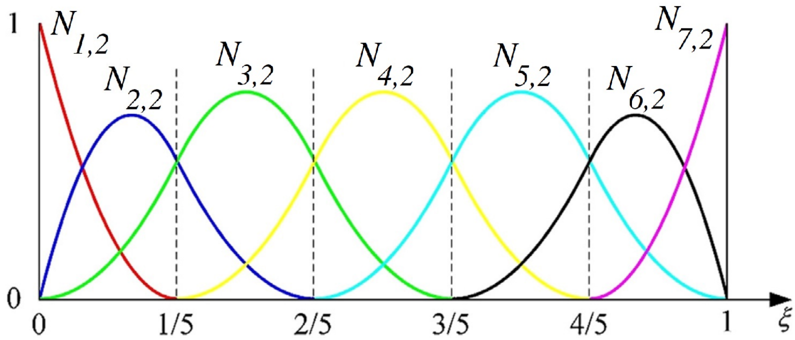


Figure 3.2. Quadratic B-spline basis functions $(p = 2, n = 7)$ **with the knot vector** $\Xi = \{0, 0, 0, 0.2, 0.4, 0.6, 0.8, 1, 1, 1\}$

$$\rho(\xi,\eta) = \sum_{i=1}^{n}\sum_{j=1}^{m} N_{i,p}(\xi)M_{j,q}(\eta)\rho_{i,j} \tag{3.12}$$

$$\mathbf{u}(\xi,\eta) = \sum_{i=1}^{n}\sum_{j=1}^{m} N_{i,p}(\xi)M_{j,q}(\eta)\mathbf{u}_{i,j} \tag{3.13}$$

In the three-dimensional space:

$$\rho(\xi,\eta,\zeta) = \sum_{i=1}^{n}\sum_{j=1}^{m}\sum_{k=1}^{l} N_{i,p}(\xi)M_{j,q}(\eta)L_{k,r}(\zeta)\rho_{i,j,k} \tag{3.14}$$

$$\mathbf{u}(\xi,\eta,\zeta) = \sum_{i=1}^{n}\sum_{j=1}^{m}\sum_{k=1}^{l} N_{i,p}(\xi)M_{j,q}(\eta)L_{k,r}(\zeta)\mathbf{u}_{i,j,k} \tag{3.15}$$

3.4. Multiregion approach

The formulation of the B-spline surfaces and solids in its parametric space, that is known as patch, has been developed. This parametric space is a square in the two-dimensional space and a cube in the three-dimensional one of unitary side. Therefore, if only one patch is considered in the definition of the problem it will be only possible to solve problems whose domain can be converted in a square or a cube by using a change of variables. This circumstance supposes an important limitation since the Finite Element Method can be used to solve problems with extremely complex domains.

In this section, it will be considered the possibility of solving problems with more complex geometries by using the Isogeometric Analysis with several patches. First, it will be necessary to analyze if it is required a certain compatibility in the definition of the patches, since one of the objectives of using Isogeometric Analysis to solve the topology optimization problem is to obtain solutions whose material layout discretization will be continuous in all the domain. This continuity means that the B-spline surfaces and solids will have to be continuous in the contact zone between two adjacent patches. As a consequence, it will not be sufficient to divide the domain in parts that will be equivalent with squares or cubes, it will be also necessary that the edge or the surface of contact between two adjacent patches will be of the same size in both patches and will be divided in the same number of knot spans. This circumstance can be seen in figure 3.3.

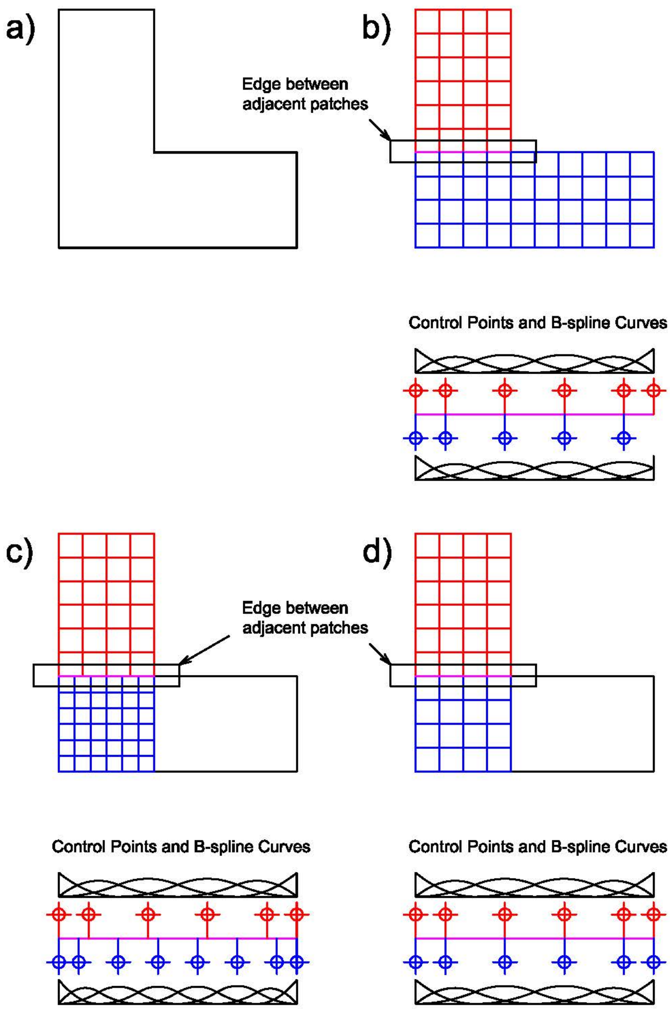


Figure 3.3. 2D multiregion approach. continuity analysis of B-spline curves in the direction of the contact between two adjacent patches. a) original domain. b) patches with different size of the edge of contact. (discontinuous) c) adjacent patches with different number of knot spans. (discontinuous) d) adjacent patches with the same size of the edge of contact and the same number of knot spans. (continuous)

Once the continuity of the B-spline surfaces and solids has been ensured between adjacent patches, the next step will be to analyze the continuity degree of these B-splines in the point of contact between both patches. This analysis will be made simultaneously for B-spline surfaces and solids, since it will be only necessary to test the continuity of the B-spline curve of the perpendicular direction to the contact between both patches. In other words, it will be necessary to analyze the continuity of only two B-splines curves, since the B-spline surfaces and solids have been obtained as the multiplication of the B-spline curve of each spatial dimension. And the continuity of the B-splines curves whose definition dimension coincides with the edge or is contained in the surface that define the contact between patches has been previously analyzed. This circumstance can be seen in figure 3.4.

The equations of the B-spline curves in the perpendicular direction to the contact between two adjacent patches that do not take a null value in the nearest knot span to this contact have been obtained as of (3.11). These formulas are:

$$\begin{aligned}
&\text{Knot Span A}\begin{cases} N_{n_A,2} = \dfrac{n_A^2}{2}\left(1-\xi_A\right)^2 \\ N_{n_A+1,2} = \dfrac{n_A^2}{2}\left(\dfrac{4}{n_A} - 3 + \xi_A\left(6 - \dfrac{4}{n_A}\right) - 3\xi_A^2\right) \\ N_{n_A+2,2} = n_A^2\left(\xi_A - \dfrac{n_A-1}{n_A}\right)^2 \end{cases} \\
&\text{Knot Span B}\begin{cases} N_{1,2} = n_B^2\left(\dfrac{1}{n_B} - \xi_B\right)^2 \\ N_{2,2} = \dfrac{n_B^2}{2}\left(\dfrac{4\xi_B}{n_B} - 3\xi_B^2\right) \\ N_{3,2} = \dfrac{n_B^2}{2}\xi_B^2 \end{cases}
\end{aligned} \tag{3.16}$$

where n_A and n_B are the number of the divisions of both patches in the perpendicular direction respectively and ξ_A and ξ_B are the value of the local coordinates in this direction in each patch.

In the continuity analysis of the B-spline curves in the perpendicular direction to the contact between two adjacent patches, the value of the B-spline curve $N_{n+2,2}$ in the Knot Span A and the B-spline curve $N_{1,2}$ in the Knot Span B and all their derivatives will have to coincide at the point of contact between both patches. This circumstance can be explained, since just the control point whose influence is defined by means of these B-spline curves is common to both patches. On the other hand, the value of the rest of the B-spline curves and all their derivatives will have to be equal to 0, since the rest of control points do not have influence in the other knot span. In other words, the B-spline curve of each control point will be equal to zero if an extrapolation of it was made in the other knot span.

For the analysis of the C^0 continuity it will be only necessary to compare the value of the B-spline curves when $\xi_A = 1$ and $\xi_B = 0$:

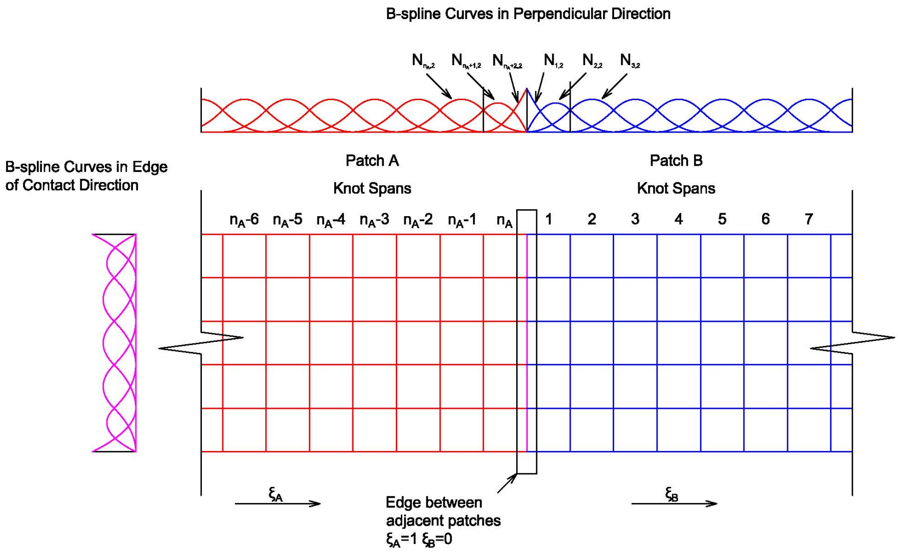


Figure 3.4. 2D multiregion approach. continuity analysis of B-spline curves in the perpendicular direction to the contact between two adjacent patches.

$$
\begin{aligned}
N_{n_A,2} = N_{n_A+1,2} = &N_{2,2} = N_{3,2} = 0 \\
N_{n_A+2,2} = &N_{1,2} = 1
\end{aligned}
\tag{3.17}
$$

Therefore, the B-spline curves are at least C^0 continuous. The next step is to calculate the derivative of this B-spline curves with respect to the parameters ξ_A and ξ_B respectively.

$$
\begin{aligned}
&\text{Knot Span A}
\begin{cases}
N'_{n_A,2} = -n_A^2\,(1-\xi_A) \\
N'_{n_A+1,2} = \dfrac{n_A^2}{2}\left(6 - \dfrac{4}{n_A} - 6\xi_A\right) \\
N'_{n_A+2,2} = 2n_A^2\left(\xi_A - \dfrac{n_A-1}{n_A}\right)
\end{cases} \\
&\text{Knot Span B}
\begin{cases}
N'_{1,2} = -2n_B^2\left(\dfrac{1}{n_B} - \xi_B\right) \\
N'_{2,2} = \dfrac{n_B^2}{2}\left(\dfrac{4}{n_B} - 6\xi_B\right) \\
N'_{3,2} = n_B^2\xi_B
\end{cases}
\end{aligned}
\tag{3.18}
$$

In the same way, for the analysis of the C^1 continuity it will be only necessary to compare the value of the first order derivative of this B-spline curves when $\xi_A = 1$ and $\xi_B = 0$:

$$
\begin{aligned}
N'_{n_A,2} &= N'_{1,2} = 0 \\
N'_{n_A+1,2} &= -2n_A \\
N'_{n_A+2,2} &= 2n_A \\
N'_{1,2} &= -2n_B \\
N'_{2,2} &= 2n_B
\end{aligned} \tag{3.19}
$$

The value of the first order derivatives of the B-spline curves are not continuous since the conditions previously considered are not fulfilled. As a consequence, the continuity of the B-spline surfaces and solids is only C^0 in the contact area between two adjacent patches, as it can be seen in figure 3.4.

Although it could be desirable to maintain the same degree of continuity in all the domain, the Multiregion Approach developed does not incorporate it when multiple patches are used. However, it will be important to analyze if the lack of continuity has an important influence in the results obtained with this Multiregion Approach. On the other hand, it is important to consider the benefits obtained by using the Multiregion Approach since the application area of Isogeometric Analysis by using it to solve the topology optimization problem has been considerably extended in terms of the domains geometry.

3.5. Overview

The choice of calculation method is a decision that have to be taken carefully, because it will have a lot of influence in the rest of the process. Finite Element Method was considered, in this thesis, in order to check the Damage Approach method and the quality of the results was good, as it was expected. However, Isogeometric Analysis has been also proposed in order to improve the accuracy of the domain's definition and reduce the CPU time, especially in complex 3D cases.

In the first place, isogeometric analysis was only introduced to do the structural analysis, with the objective of diminishing the amount of time that have to be used to do it. The result was that the CPU time employed for solving the structural analysis with the Isogeometric Analysis was lower than with the Finite Element Method. This CPU time reduction can be explained because of the lower number of points used to do it with the Isogeometric Analysis. This circumstance will be justified in following chapters.

On the other hand, Isogeometric Analysis will be also employed in the design domain's definition in order to homogenize the process. Domain's definition will be improved, because the limitation of the domain that is imposed by finite element method is overtaken without the need to diminish the size of the domain's elements.

Finally, the Multiregion Approach has been introduced in this thesis, since the application area of the Isogeometric Analysis on its own is reduced to the solution of problems whose domain can be defined by using only one square or one cube. Therefore, the introduction of this approach in the formulation will allow to solve problems whose domain will be not a square or a cube by using the Isogeometric Analysis.

Once one alternative method of the Finite Element Method has been introduced, the Isogeometric Analysis, the next step will be to formulate the topology optimization problem. In this thesis this formulation will be made for both methods, what will make possible to do a comparative analysis between both methods later.

CHAPTER 4

Structural analysis

"There is no branch of mathematics, however abstract, which may not some day be applied to phenomena on the real world"
Nikolai Lobachevsky, (1792-1856).

4.1. Introduction

The topology optimization problem that will be solved in this thesis consist of two parts: the first one, the structural analysis, and the second one, the optimization problem. The structural analysis module must be very accurate and require small computer requirements since it will be computed at each optimization iteration.

Furthermore, the structural analysis should have the possibility of incorporating design variables in this process, like the relative density that will have influence in the structural analysis. Moreover, the continuous character of the structural displacements and structural stresses in the domain should be taken into consideration when the procedure will be defined.

For all these reasons, the finite element method has been firstly used for the structural analysis. In this formulation, the most common approach consists on assigning each design variable at each element of the mesh. As a result of this, the value of the relative density in each element will be constant and it will match with the value of the design variable. However, the results of the topology optimization problem have been good enough, considering the way material layout is defined. Despite of this, other alternative ways to establish the material layout in the domain has been proposed, and consequently used. Nevertheless, in the finite element method, the use of these alternatives increase the complexity of the problem, since the number of design variables have to be increased considerably.

Instead of this, the isogeometric analysis has been proposed as alternative not only due to the fact that less design variables have to be used to obtain a better definition of the domain than with finite element method, but also the fact that results that have been obtained with the structural analysis have the possibility of being continuous in all the domain with its derivatives also continuous. On the other hand, the finite element method can be replicated with the isogeometric analysis, if the knot vector is defined properly. Nonetheless, this possibility is not recommended, since the serendipity elements can not be replicated.

In order to check all these aspects, a comparative analysis between structural analysis with finite element method and isogeometric analysis will be shown in chapter 9, when both alternatives will be used to solve some topology optimization problems.

Finally, the structural analysis model, that will be proposed in this thesis, is a standard approach of the finite element method ([Hughes, 2000], [Navarrina et al., 2005], [Oñate, 1995], [Zienkiewickz & Taylor, 2004]). At this point, it is important to take into consideration the effect of the design variables. The incorporation of the design variables makes necessary to introduce some changes in the typical formulation. Because as it was mentioned previously, the design variables are the basis to define the material layout in the domain, and this circumstance will have to be considered in the structural analysis.

To conclude, the structure of the chapter will be: firstly, formulation of structural analysis without design variables will be developed. Secondly, design variables will be introduced in this analysis. This structure will be developed not only in the two-dimensional space but also in the three-dimensional space. Finally, as it was commented in Section 3.3., the method developed in this chapter, can be used with both methods: the finite element method and the isogeometric analysis.

4.2. Formulation of structural problem

Given a certain region Ω^0 of the space filled by a determined object on that certain external loads will be applied. As a result of this loads, this object will be deformed and consequently another different region Ω will be filled.

In the first place, it will be possible to establish a connection between the location of each point at Ω^0 and the position that this point will have in Ω due to the strain.

In case of defining the position of each point in both situations with respect to the same point in the space, the displacement $\mathbf{u}$ of each point can be defined as:

$$\mathbf{u}(\mathbf{r}^0) = \mathbf{r}(\mathbf{r}^0) - \mathbf{r}^0 \tag{4.1}$$

where $\mathbf{r}$ and $\mathbf{r}^0$ are the coordinates of a certain point in Ω and Ω^0 respectively.

Once displacements are calculated, strains $\boldsymbol{\varepsilon}$ and stresses $\boldsymbol{\sigma}$ in each point of the domain, can be calculated. In order to apply the classical structural analysis methods, several hypotheses have to be considered. These hypotheses are:

- Small displacements
- Small displacements derivative

- Elastic and linear material

Hereinafter, $\mathbf{L}$ will be the strain-displacement matrix, and $\mathbf{D}$ will be the constitutive matrix of the material. On the other hand, $\boldsymbol{\varepsilon}$ and $\boldsymbol{\sigma}$ will be respectively, the strain and stress vectors in a certain point, [Hughes, 2000]. Thus,

$$\boldsymbol{\varepsilon} = \mathbf{L}\mathbf{u} \qquad\qquad \boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon} \tag{4.2}$$

Moreover, there are several ways of establishing loads in the structural analysis. It can be external loads applied for unit of volume $\mathbf{b}(\mathbf{r}^0)$ over the body in the domain, and loads applied for unit of surface $\mathbf{t}(\mathbf{r}^0)$ over the boundary of the domain Γ^0_σ.

Once the loads are imposed, the structural analysis consist on:

$$\begin{array}{rll}
\text{Calculate} & \mathbf{u} \in H_u & \\
\text{with} & a(\mathbf{w},\mathbf{u}) = (\mathbf{w},\mathbf{b})_{\Omega^0} + (\mathbf{w},\mathbf{t})_{\Gamma^0_\sigma} & \forall \mathbf{w} \in \mathbf{H_w} \\
\text{being} & a(\mathbf{w},\mathbf{u}) = \displaystyle\iiint_{\Omega^0} (\mathbf{L}\mathbf{w})^T \mathbf{D}(\mathbf{L}\mathbf{u}) d\Omega & \\
& (\mathbf{w},\mathbf{b})_{\Omega^0} = \displaystyle\iiint_{\Omega^0} \mathbf{w}^T \mathbf{b} d\Omega & \\
& (\mathbf{w},\mathbf{t})_{\Gamma^0_\sigma} = \displaystyle\iint_{\Gamma^0_\sigma} \mathbf{w}^T \mathbf{t} d\Gamma &
\end{array} \tag{4.3}$$

where H_u are the trial functions and H_w are the test functions.

It is also important to take into consideration that these functions have to satisfy no-essential boundary conditions and homogeneous boundary conditions, respectively. [Hughes, 2000], [Zienkiewickz & Taylor, 2004], [Navarrina et al., 2005].

4.2.1. Numerical model

Once the structural analysis formulation has been established in (4.3), it is important to remark that the analytical resolution of this problem will not be always possible, and in the case that it will be possible, this can involve an arduous task.

With the objective of making this procedure simple, exact solution of the problem will be approximated by a finite discretization of the domain. This approximation can be made either with the Finite Element Method or the Isogeometric Analysis. As it was established in Chapter 3, both methods will have the same formulation, with the exception of some aspects, also mentioned.

The numerical solutions of the structural problem will require to do several steps. Firstly, it will be necessary to replace the continuous space of functions H_u and H_w with its equivalent discretized spaces H_u^h and H_w^h. Moreover, trial and test functions will have to be also replaced with $\mathbf{u}^h$ and $\mathbf{w}^h$ respectively.

On the other hand, it will be also necessary to choose thoughtfully a basis of the trial and test functions in subspaces H_u^h and H_w^h, that will be symbolized in the formulation as $\left\{\Phi_i\left(\mathbf{r}^0\right)\right\}$ and $\left\{w_j\left(\mathbf{r}^0\right)\right\}$ respectively. Furthermore, it will be also required to assure that homogeneous boundary conditions will be satisfied.

Once all the previous steps have been made the approximated solution of the structural problem that has been proposed in the previous sections will be:

$$\begin{aligned} \mathbf{u}^h(\mathbf{r}^0) &= \mathbf{u}^p(\mathbf{r}^0) + \sum_{i=1}^{N} \mathbf{\Phi}_i(r^0)\boldsymbol{\alpha}_i & \mathbf{\Phi}_i(\mathbf{r}^0) &= \phi_i(\mathbf{r}^0)\mathbf{I}_3 \\ \mathbf{w}^h(\mathbf{r}^0) &= \sum_{j=1}^{N} \mathbf{W}_j(\mathbf{r}^0)\boldsymbol{\beta}_j & \mathbf{W}_j(\mathbf{r}^0) &= w_j(\mathbf{r}^0)\mathbf{I}_3 \end{aligned} \tag{4.4}$$

where $\boldsymbol{\alpha}_i|_{i=1,\ldots,N}$ is the unknown vector and $\mathbf{u}^h(\mathbf{r}^0)$ is the displacement vector in the point $\mathbf{r}^0$ of the domain.

Furthermore, it should be noted that unknowns will have different physical meaning, if the Finite Element Method or the Isogeometric Analysis is used, since the unknowns will represent nodal displacements or control points displacements respectively.

When spatial discretization of the domain has to be made, it is essential that next conditions are respected.

$$\Omega^0 = \bigcup_{e=1}^{N_e} \Omega_e \qquad \Omega_i \bigcap \Omega_j = \phi \qquad \forall i \neq j \tag{4.5}$$

where in case of using Finite Element Method Ω_e will be the part of the domain that represents the element e and N_e is the number of elements in the mesh, while in case of Isogeometric Analysis, Ω_e will be the part of the domain that fills the knot span e and N_e is the number of knot spans in the mesh.

The trial and test functions will be usually formulated through Galerkin Isoparametric Functions in the structural analysis problems, and for this reason:

$$w_j(\mathbf{r}^0) = \phi_j(\mathbf{r}^0) \tag{4.6}$$

As a result of this consideration, Finite Element model or Isogeometric model with Galerkin approximations consist on:

$$\begin{aligned} &\text{Calculate} && \boldsymbol{\alpha} = \{\boldsymbol{\alpha}_i\} && i = 1, ..., N \\ &\text{with} && \sum_{i=1}^{N} \mathbf{K}_{ji}\boldsymbol{\alpha}_i = \mathbf{f}_j \quad j = 1, ..., N \\ &\text{being} && \mathbf{K}_{ji} = a\left(\mathbf{\Phi}_j, \mathbf{\Phi}_i\right) \\ &&& \mathbf{f}_j = \left(\mathbf{\Phi}_j, \mathbf{b}\right)_{\Omega^0} + \left(\mathbf{\Phi}_j, \mathbf{t}\right)_{\Gamma^0_\sigma} + a\left(\mathbf{\Phi}_j, \mathbf{u}^p\right) \end{aligned} \tag{4.7}$$

where $\mathbf{K}_{ji}$ and $\mathbf{f}_j$ will have to be obtained by assembling elemental contributions of each element or knot span, as:

$$\mathbf{K}_{ji} = \sum_{e=1}^{N_e} \mathbf{K}_{ji}^e \qquad \mathbf{f}_j = \iint_{\Gamma_\sigma^0} \boldsymbol{\Phi}_j^T \mathbf{t} d\Gamma + \sum_{e=1}^{N_e} \mathbf{f}_j^e \tag{4.8}$$

where

$$\begin{aligned} \mathbf{K}_{ji}^e &= \iiint_{\Omega_e} (\mathbf{L}\boldsymbol{\Phi}_j)\, \mathbf{D}\, (\mathbf{L}\boldsymbol{\Phi}_i)\, d\Omega \\ \mathbf{f}_j^e &= \iiint_{\Omega_e} \left(\boldsymbol{\Phi}_j^T \mathbf{b} - (\mathbf{L}\boldsymbol{\Phi}_j)\, \mathbf{D}\, (\mathbf{L}\mathbf{u}^p)\right) d\Omega \end{aligned} \tag{4.9}$$

Once the structural analysis has been numerically established, it will be necessary to define the differential operator and the constitutive matrix for all the cases that will be object of analysis in this thesis, considering small displacement gradients.

For two-dimensional problems differential operator $\mathbf{L}$ can be defined as:

$$\mathbf{L} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \tag{4.10}$$

and in the case of three-dimensional problems the matrix $\mathbf{L}$ will be:

$$\mathbf{L} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix} \tag{4.11}$$

In the same way, constitutive matrix $\mathbf{D}$ will be defined in two-dimensional problems for elastic, linear and isotropic materials as:

$$\mathbf{D} = \begin{bmatrix} d_{11} & d_{12} & 0 \\ d_{21} & d_{22} & 0 \\ 0 & 0 & d_{33} \end{bmatrix} \tag{4.12}$$

where the value of the matrix elements will be different depending on the nature of the problem:

$$\begin{array}{cc} \text{Plane Stress} & \text{Plane Strain} \\ d_{11} = d_{22} = \dfrac{E}{1-\nu^2} & d_{11} = d_{22} = \dfrac{E(1-\nu)}{(1+\nu)(1-2\nu)} \\ d_{12} = d_{21} = \dfrac{\nu E}{1-\nu^2} & d_{12} = d_{21} = \dfrac{\nu E}{(1+\nu)(1-2\nu)} \\ d_{33} = \dfrac{E}{2(1+\nu)} = G & d_{33} = \dfrac{E}{2(1+\nu)} = G \end{array} \tag{4.13}$$

and in three-dimensional problems $\mathbf{D}$ will be defined as:

$$\mathbf{D} = \begin{bmatrix} d_{11} & d_{12} & d_{13} & 0 & 0 & 0 \\ d_{21} & d_{22} & d_{23} & 0 & 0 & 0 \\ d_{31} & d_{32} & d_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & d_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & d_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & d_{66} \end{bmatrix} \tag{4.14}$$

and the value of the matrix elements will be:

$$\begin{aligned} d_{11} = d_{22} = d_{33} &= \frac{(1-\nu)E}{(1+\nu)(1-2\nu)} \\ d_{12} = d_{21} = d_{13} = d_{31} = d_{23} = d_{32} &= \frac{\nu E}{(1+\nu)(1-2\nu)} \\ d_{44} = d_{55} = d_{66} &= \frac{E}{2(1+\nu)} = G \end{aligned} \tag{4.15}$$

In matrix elements, E is the Young modulus of the material and ν the Poisson coefficient [Oñate, 1995], [Hughes, 2000], [Zienkiewickz & Taylor, 2004], [Navarrina et al., 2005].

Once the solution of the numerical structural problem (4.7) is obtained, an approximation of displacements, strains and stresses through (4.4) and (4.2) can be calculated as:

$$\begin{aligned} &\mathbf{u}^h(\mathbf{r}^0) = \mathbf{u}^p(\mathbf{r}^0) + \sum_{i=1}^{N} \mathbf{\Phi}_i(r^0)\boldsymbol{\alpha}_i \\ &\boldsymbol{\varepsilon}^h(r^0) = \mathbf{L}\mathbf{u}^h(\mathbf{r}^0) \\ &\boldsymbol{\sigma}^h(r^0) = \mathbf{D}\boldsymbol{\varepsilon}^h(\mathbf{r}^0) \end{aligned} \tag{4.16}$$

4.3. Structural analysis with Isogeometric Analysis

Once the numerical structural analysis has been proposed in the previous section, the next step is to introduce the relative density by means of the design variables of the topology optimization problem in the formulation previously developed. For this purpose, a similar scheme of the numerical structural analysis will be used.

In the first place, it will be necessary to take into consideration that the domain will be filled by a porous material. The design variable that will be represented as $\rho(\mathbf{r}^0)$ define the relative density in each point of the domain. This variable can take whatever value between 0, that means lack of material, and 1, that indicates solid material.

Although there are different alternatives to define the material layout in the domain with both methods: Finite Element Method and Isogeometric Analysis, the approaches that will be used in this thesis will be described now. In Finite Element Method, the value of density in each element will match with the value of the design variable associated to this element. In the Isogeometric Analysis, it will be necessary to use (3.14) to calculate density in whatever point of the domain as of the design variables. In addition, it is not advisable to use different discretizations to make the structural analysis and the material layout with the Isogeometric Analysis. As a consequence, in both methods, the relative density can be expressed as a function of the design variables.

According to previous sections, structural analysis has as objective the calculation of the displacement distribution in the domain. Once this displacement distribution is calculated, strains and stresses can be computed for whatever material distribution in the domain.

If a differential region of material around P^0 is defined as $d\Omega$, the volume of material in this region can be easily calculated as $\rho(\mathbf{r}^0)d\Omega$.

Moreover, the design variable can be stated in the structural analysis as:

$$\begin{aligned}
\text{Given} \quad & \rho(\Omega^0) \\
\text{Calculate} \quad & \mathbf{u} \in H_u \\
\text{with} \quad & a(\mathbf{w},\mathbf{u}) = (\mathbf{w},\mathbf{b})_{\Omega^0} + (\mathbf{w},\mathbf{t})_{\Gamma^0_\sigma} \qquad \forall \mathbf{w} \in \mathbf{H_w} \\
\text{being} \quad & a(\mathbf{w},\mathbf{u}) = \iiint_{\Omega^0} (\mathbf{Lw})^T \mathbf{D}(\mathbf{Lu})\rho d\Omega \\
& (\mathbf{w},\mathbf{b})_{\Omega^0} = \iiint_{\Omega^0} \mathbf{w}^T \mathbf{b} \rho d\Omega \\
& (\mathbf{w},\mathbf{t})_{\Gamma^0_\sigma} = \iint_{\Gamma^0_\sigma} \mathbf{w}^T \mathbf{t} d\Gamma
\end{aligned} \tag{4.17}$$

where $\mathbf{b}$ represent the external loads applied for unit of volume over the body in the domain and $\mathbf{t}$ represent the loads applied for unit of surface over the boundary of the domain.

To conclude with this section, a comparative analysis between the structural analysis with design variables and without them can be made. The formulation of both problems can be watched in (4.17) and (4.3) respectively. As a result, the only difference between both methods is the appearance of the design variables in the integration of volume elemental terms.

On the other hand, in the same way that elemental contributions have been calculated, strains and stresses can be also calculated by means of the same equations (4.16).

However, the design variable can have a null value that makes that this analysis does not have sense, because the volume will be also zero, and displacements, strains and stresses will not exist. This will be solved in the formulation of the optimization problem through the establishment of an inferior limit of the design variables slightly higher than 0. This will also solve problems with matrixes in the calculations of structural analysis, since the matrix obtained will not be singular. This circumstance can not be ensured if the design variables can be equal to zero.

4.3.1. Numerical model with Isogeometric definition of density

The approach considered to define the relative density in the domain with the Isogeometric Analysis introduce a little drawback. This circumstance makes necessary to propose an approximated solution of the structural analysis. On the other hand, it is important that the same discretization scheme will be used not only for the structural analysis but also for the definition of the material layout because of the continuity properties of the isogeometric analysis.

At this point, it is important to take into consideration the differences between the use of finite element method and isogeometric analysis for the definition of material layout.

Although more complex definition of the material distribution can be made, it is very usual in the finite element method, to define the relative density as a constant value in the domain of each element of the mesh ([Cheng & Jiang, 1992], [Duysinx & Sigmund, 1998], [Pereira et al., 2004], [Sigmund, 1994], among others.

Even though it could be possible to use different schemes to make the structural analysis and to define the material layout in case of Isogeometric Analysis, this possibility is not advisable. This approach has been tested in this research by means of the definition of the density in each knot span as a constant value. However, the results of the structural analysis obtained with this approach were different in contrast to the results obtained with the Finite Element Method for the same problem.

As it was previously commented, it will be necessary that there is a coincidence between the discretization scheme used to do the structural analysis and to define the material distribution. The use of different discretization schemes introduces a difference in the continuity degree between the displacements field and the material layout. This can suppose that the global stiffness matrix, that depends on the material layout, used to solve the structural analysis and the displacements field obtained with the solution of this structural analysis are formulated with non-coherent discretization schemes. This circumstance happens in the case previously commented. For this reason, this procedure should not be adopted in isogeometric analysis.

As a result of this, isogeometric analysis will be maintained to make structural analysis and the same discretization scheme will be implemented for defining the material layout. This situation does not suppose a big effort, since the shape functions used in both cases are the same. In other words, the relative density ρ will be defined with the same formula that the structural displacement $\mathbf{u}$. In both case, the relative density and the structural displacement will be a linear combination of their values in the control points.

Density in control points in isogeometric analysis as well as density in elements in case of finite element method will be the design variables of the topology optimization problem. So when ρ is known, the problem that will be necessary to solve is:

$$\begin{aligned} &\text{Calculate} \qquad \boldsymbol{\alpha}(\rho) \\ &\text{with} \qquad \sum_{i=1}^{N} \mathbf{K}_{ji}(\rho)\boldsymbol{\alpha}_i(\rho) = \mathbf{f}_j(\rho) \qquad j = 1, ..., N \end{aligned} \tag{4.18}$$

All terms in (4.18) can be also calculated, as usual, through the assembly of the independent contribution of each knot span as:

$$\begin{aligned} \mathbf{K}_{ji}(\rho) &= \sum_{e=1}^{N_e} \mathbf{K}_{ji}^{e}(\rho) \\ \mathbf{f}_j(\rho) &= \iint_{\Gamma_\sigma^0} \boldsymbol{\Phi}_j^T \mathbf{t} d\Gamma + \sum_{e=1}^{N_e} \mathbf{f}_j^e(\rho) \end{aligned} \tag{4.19}$$

where the elemental contributions will be:

$$\begin{aligned} \mathbf{K}_{ji}^{e}(\rho) &= \iiint_{\Omega_e} (\mathbf{L}\boldsymbol{\Phi}_j)^T \mathbf{D}(\mathbf{L}\boldsymbol{\Phi}_i)\rho d\Omega \\ \mathbf{f}_j^e(\rho) &= \iiint_{\Omega_e} (\boldsymbol{\Phi}_j^T \mathbf{b} - (\mathbf{L}\boldsymbol{\Phi}_j)^T \mathbf{D}(\mathbf{L}\mathbf{u}^p))\rho d\Omega \end{aligned} \tag{4.20}$$

Once the solution $\boldsymbol{\alpha}(\rho)$ of the problem (4.18) has been obtained, the next step is to calculate the displacements field. However, only an approximation of the displacements field will be obtained. In the same way that in the previous cases, once the value of the displacements is known in a certain point, strains and stresses in this point can be calculated. For this purpose, it will be necessary to use the next equations.

$$\begin{aligned} \mathbf{u}^h(\mathbf{r}^0, \rho) &= \mathbf{u}^p(\mathbf{r}^0) + \sum_{i=1}^{N} \boldsymbol{\Phi}_i(r^0)\boldsymbol{\alpha}_i(\rho) \\ \boldsymbol{\varepsilon}^h(\mathbf{r}^0, \rho) &= \mathbf{L}\mathbf{u}^h(\mathbf{r}^0, \rho) \\ \boldsymbol{\sigma}^h(\mathbf{r}^0, \rho) &= \mathbf{D}\boldsymbol{\varepsilon}^h(\mathbf{r}^0, \rho) \end{aligned} \tag{4.21}$$

The displacements in the control points are obtained with the resolution of structural problem. The value of the displacements in whatever point of the domain have to be calculated by means of the use of the shape functions. Nevertheless, strains and stresses can be calculated as usual, since they do not depend on the relative density. For this reason, it will be possible to calculate independently strains and stresses when displacements are known.

Apart from this, it is important to comment that a procedure whose objective is to provide empty-full solutions has been proposed. From the point of view of industry these solutions are more interesting since they require an easier and cheaper manufacturing procedure.

This method that has been proposed by [Navarrina et al., 2005] and [París, 2007], has a similar approach to SIMP model, procedure that has been firstly proposed by [Bendsøe, 1989]. Furthermore, this method will be also used in this thesis.

4.4. Overview

Structural Analysis has been formulated with a procedure that meets several requirements no matter whether finite element method or isogeometric analysis is used. This method has to be reliable, effective and in case of being possible, it will solve this analysis quickly, because during the resolution of topology optimization problem many designs will have to be analyzed.

In this thesis, continuous structural problems will have to be settled. Although finite element method could be used to do it, Isogeometric analysis will be used instead of this, because of the improvement obtained regarding to the CPU time. This circumstance can be easily explained, since the number of points used for the structural analysis with the Isogeometric Analysis is considerably lower than with the Finite Element Method.

Furthermore, in the same way that in structural analysis, Finite Element Method has also an important restriction to define the material distribution in the domain, since the number of design variables used to obtain comparable solutions is also higher. For this reason, the use of the Isogeometric Analysis is advisable if the purpose is obtaining solutions with high spatial definition.

The use of Isogeometric Analysis instead of Finite Element Method for the density distribution, is not a big issue, since the same numerical code can be used in both methods. The only thing that have to be considered is the fact that the results obtained with each method have a different meaning as it was commented in Chapter 3.

Moreover, a method in that a relationship between elemental stiffness matrix and density is established has been developed. For all these reasons, it will be possible to use the finite element formulation, considering the changes that Isogeometric Analysis introduces regarding to Finite Element Method.

Therefore, it has been only necessary to include the density effect in finite element formulations, that supposes an improvement in the domains definition and also a complication in solving numerical problem, due to the effect that a non-constant value of density has in the integration.

To conclude, once structural analysis method through Isogeometric Analysis has been established, the next step is to define the other part of the structural topology optimization algorithm: The Optimization Problem.

Chapter 5

Optimization problem

"We cannot solve our problems with the same thinking we used when we created them"
Albert Einstein, (1879-1955).

5.1. Introduction

The Topology Optimization problem can be formulated in different ways. This formulation is related to the different definitions of the material layout. This material distribution can have discontinuities in some parts of the domain or can be completely continuous. Moreover, the formulation of the problem can be made with the use of discrete design variables or continuous design variables.

On the one hand, the use of discrete design variables in the topology optimization problem has certain limitations. The main restriction is the impossibility of using the traditional optimization algorithms, since the design variables can only take a certain group of values. Consequently, the solution of this problem could require the analysis of all the possible combinations in order to obtain the optimum solution.

On the other hand, the use of continuous design variables has been more widespread. This circumstance is due to the use of the traditional optimization algorithms. These algorithms give the optimum solution without the necessity of checking all the possible combinations.

Therefore, a topology optimization problem of structures with the use of the Finite Element Method and the Isogeometric Analysis by means of continuous design variables will be developed in this thesis. Although, this choice has been motivated for the possibility of using the classical optimization algorithms, the correct problem will be the binaries problem (0-1).

The binaries problem only requires to establish the parts of the domain where the material has to be placed. This formulation is ideal from the manufacturing point of view, since the manufacturing techniques such as three-dimensional printing only allow to manufacture this kind of solutions. In other words, this circumstance is the main drawback of the use of continuous design variables, since it is impossible to obtain this manufacturing material distribution.

Additionally, as it was previously mentioned, it will be important to take into consideration the relationship between the design variable and the relative density. This relationship will depend on the method used to define the material layout in the domain.

The Finite Element Method proposed in this thesis only ensures the continuity in the value of the design variables, since each design variable will define the material distribution in each element of the mesh by means of a constant value per element. For this reason, the value of the design variable and the relative density will coincide in all the points of the domain, however, there will be discontinuities in the value of the relative density between adjacent elements.

On the contrary, the Isogeometric Analysis proposed ensures the continuity not only in the value of the design variables but also in the relative density in all the points of the domain. Independently of the method that will be used for defining the material distribution in the domain, design variables will be able to take only values between 0 and 1.

For all this reasons, the Isogeometric Analysis has been finally chosen to define the material distribution, because of the quality of the results that can be obtained. In this method, density in each point of the domain will have to be calculated as a linear combination of design variables (3.14).

Moreover, it will be important to take into consideration that with the exception of interpolatory points, design variables will not have physical meaning since there is no coincidence between relative density and design variables, as it was explained in Chapter 3.

Finally, continuous design variables will allow to do the sensitivity analysis of the structural analysis previously developed in Chapter 4, as displacement derivatives will have to be calculated. Because of this, typical derivative techniques and optimization algorithms with continuous design variables can be also used for solving the resulting optimization problem.

In this chapter, the minimum weight general problem with stress constraints will be firstly formulated. Hereafter the way of establishing objective function will be analyzed and finally the Damage Approach will be developed to consider the effect of all stress constraints.

5.2. Minimum weight formulation

Generally, whatever optimization problem can be formulated as:

$$\begin{array}{lll} \text{Calculate} & \boldsymbol{\rho} = \{\rho_i\} & i = 1, ..., n \\ \text{ that minimizes} & F(\boldsymbol{\rho}) & \\ \text{verifying} & g_j(\mathbf{r}_j^0, \boldsymbol{\rho}) \leq 0 & j = 1, ..., m \\ & h_l(\mathbf{r}_l^0, \boldsymbol{\rho}) = 0 & l = 1, ..., p \\ & \rho_{min} \leq \rho_i \leq \rho_{max} & i = 1, ..., n \end{array} \tag{5.1}$$

where $\boldsymbol{\rho} = \{\rho_i\}$ is the design variable vector, $F(\boldsymbol{\rho})$ is the objective function to be minimized, g_j are the inequality constraints of the problem and, finally, h_l are the equality constraints of the problem.

Furthermore, in the optimization problem n is the number of design variables, while m and p are respectively the number of inequality and equality constraints.

Finally, as far as the optimization problem concerns, it will be also necessary to establish not only an inferior limit of the design variables, but also a superior one, that will be respectively defined as ρ_{min} and ρ_{max}, and, in other words, they will represent the side constraints of the design variables.

However, in the topology optimization problems, the general optimization problem (5.1) can be streamlined, when, for instance, equality constraints are nonexistent. The objective function will be usually the structural weight that will have to be minimized. Consequentially, the topology optimization problem of structures can be reformulated as:

$$\begin{array}{lll} \text{Calculate} & \boldsymbol{\rho} = \{\rho_i\} & i = 1, ..., n \\ \text{ that minimizes} & F(\boldsymbol{\rho}) & \\ \text{verifying} & g_j(\mathbf{r}_j^0, \boldsymbol{\rho}) \leq 0 & j = 1, ..., m \\ & \rho_{min} \leq \rho_i \leq \rho_{max} & i = 1, ..., n \end{array} \tag{5.2}$$

Additionally, in the topology optimization problem of structures, the density will be the characteristic that will be defined by the design variables. Because of this, side constraints will have to take these values:

- Superior Limit: $\rho_{max} = 1$ (space full of material)
- Inferior Limit: $\rho_{min} = 0$, (space empty of material)

Although taking $\rho_{min} = 0$ as the inferior limit would be the ideal situation, the consequent singularity problems with stiffness matrix require to take a value slightly higher than zero. For this reason, the typical value for ρ_{min} in the literature is 0.001 ([Bendsøe & Kikuchi, 1988], [Bendsøe, 1995]). In this thesis this value will be also adopted.

Once side constraints of the topology optimization problem have been established and analyzed, it will be also important to take into consideration that the key factors of topology optimization problem will be, the definition of the most suitable objective function, that will have a big influence in the results and the introduction of a damage constraint that will ensure that a certain group of individual constraints will be satisfied.

As a result of all this, the topology optimization problem will consist of only the objective function and the damage constraint, and it can be formulated as:

$$\begin{array}{lll} \text{Calculate} & \boldsymbol{\rho} = \{\rho_i\} & i = 1, ..., n \\ \text{that minimizes} & F(\boldsymbol{\rho}) & \\ \text{verifying} & g(\boldsymbol{\rho}) \leq 0 & \\ & \rho_{min} \leq \rho_i \leq \rho_{max} & i = 1, ..., n \end{array} \tag{5.3}$$

Finally, at this point, topology optimization problem is not completely formulated. The complete optimization problem will be developed and analyzed in depth in the next sections.

5.3. Objective function

As mentioned above, one of the objectives of this thesis is to design structures with minimum weight. These structures will have to be able to withstand all applied loads. For this reason, the objective function is based on the minimization of the structural weight, as:

$$F = \sum_{e=1}^{N_e} \int_{\Omega_e} \rho d\Omega_e \tag{5.4}$$

5.3.1. Uniform density per element formulation

The Finite Element Method, one of the alternatives that has been commented in previous chapters, can be used to solve the topology optimization problems of structures.

In this approach, each design variable will represent the density in each element of the mesh by means of a constant value. Therefore, the objective function of the topology optimization problem in the Finite Element Method can be expressed as:

$$F = \sum_{e=1}^{N_e} \rho_e \int_{\Omega_e} d\Omega_e \tag{5.5}$$

On the other hand, if all the elements of the finite element mesh have the same volume, this formula (5.5) can be simplified as:

$$F = \sum_{e=1}^{N_e} \rho_e V_{\Omega_e} = \sum_{e=1}^{N_e} W_e \tag{5.6}$$

5.3.2. Material density distributions by means of Isogeometric interpolation

The Isogeometric Analysis is the other alternative that can be used to solve the topology optimization problem of structures.

Unlike the Finite Element Method, in this approach, the density in each point of the domain has to be calculated through a linear combination of the design variables, as it was previously analyzed in (3.14). Consequently, it will be necessary to develop the integral of all the B-spline surfaces or the B-spline solids. Therefore, the objective function in Isogeometric Analysis can be expressed as:

$$\begin{aligned} &\text{2D} \quad F = \sum_{e=1}^{N_e}\sum_{i=1}^{n}\sum_{j=1}^{m}\int_{\Omega_e} N_{i,p}(\xi)M_{j,q}(\eta)\rho_{i,j}\ d\Omega_e \\ &\text{3D} \quad F = \sum_{e=1}^{N_e}\sum_{i=1}^{n}\sum_{j=1}^{m}\sum_{k=1}^{l}\int_{\Omega_e} N_{i,p}(\xi)M_{j,q}(\eta)L_{k,r}(\zeta)\rho_{i,j,k}\ d\Omega_e \end{aligned} \tag{5.7}$$

where n, m and l are the number of the B-spline basis function, and p, q and r are the degree of the B-spline basis function in each dimension, respectively. In this thesis, p, q and r will be always equal to 2, while n, m and l will depend on the meshing considered. On the other hand, the domain Ω_e is not defined in the parametric space of B-splines, and it will be necessary to do a change of variables.

$$\begin{aligned} &x = \xi\ l_x \qquad && y = \eta\ l_y \qquad && z = \zeta\ l_z \\ &dx = d\xi\ l_x && dy = d\eta\ l_y && dz = d\zeta\ l_z \end{aligned} \tag{5.8}$$

Therefore Ω_e will have to be replaced by:

$$\begin{aligned} &\text{2D} \quad d\Omega_e = dxdy = d\xi d\eta\ l_x l_y = d\xi d\eta\ V_{patch} = d\xi d\eta\ V_{ks}\ n_x n_y \\ &\text{3D} \quad d\Omega_e = dxdydz = d\xi d\eta d\zeta\ l_x l_y l_z = d\xi d\eta d\zeta\ V_{patch} = d\xi d\eta d\zeta\ V_{ks}\ n_x n_y n_z \end{aligned} \tag{5.9}$$

where l_x, l_y and l_z are the dimensions of the patch in each direction, V_{patch} is its volume, n_x, n_y and n_z are the number of knot spans in each direction and V_{ks} is the volume of one knot span. Moreover, as the value of all design variables is a constant, the formula (5.7) can be also reduced to:

$$\begin{aligned} &\text{2D} \quad F = \sum_{e=1}^{N_e}\sum_{i=1}^{n}\sum_{j=1}^{m}\rho_{i,j}V_{ks}n_x n_y\int_{\Omega_e} N_{i,p}(\xi)M_{j,q}(\eta)\ d\xi d\eta \\ &\text{3D} \quad F = \sum_{e=1}^{N_e}\sum_{i=1}^{n}\sum_{j=1}^{m}\sum_{k=1}^{l}\rho_{i,j,k}V_{ks}n_x n_y n_z\int_{\Omega_e} N_{i,p}(\xi)M_{j,q}(\eta)L_{k,r}(\zeta)\ d\xi d\eta d\zeta \end{aligned} \tag{5.10}$$

Knot span	1	2	3	...	i	...	n-2	n-1	n	Aggregation
$N_{1,2}$	1/3	0	0	...	0	...	0	0	0	1/3
$N_{2,2}$	1/2	1/6	0	...	0	...	0	0	0	2/3
$N_{3,2}$	1/6	2/3	1/6	...	0	...	0	0	0	1
$N_{4,2}$	0	1/6	2/3	...	0	...	0	0	0	1
...	...	...	...	...	...	...	...	...	...	...
$N_{i,2}$	0	0	0	...	1/6	...	0	0	0	1
$N_{i+1,2}$	0	0	0	...	2/3	...	0	0	0	1
$N_{i+2,2}$	0	0	0	...	1/6	...	0	0	0	1
...	...	...	...	...	...	...	...	...	...	...
$N_{n-1,2}$	0	0	0	...	0	...	2/3	1/6	0	1
$N_{n,2}$	0	0	0	...	0	...	1/6	2/3	1/6	1
$N_{n+1,2}$	0	0	0	...	0	...	0	1/6	1/2	2/3
$N_{n+2,2}$	0	0	0	...	0	...	0	0	1/3	1/3

Table 5.1. Integral solution of quadratic basis functions - 1D

Additionally, it will be possible to observe in (5.10) the fact that basis functions will have to be integrated in the domain of each knot span. For this reason, in table 5.1, the result of the B-spline curve integral in each knot span is shown.

On the other hand, in two and three dimensions, the only thing that has to be done is the replacement of the integral by the multiplication of the two or three coefficients previously calculated and the volume of the knot span. If all the knots span has the same volume, it will not be necessary to take this value into consideration for each product and can be applied one time at the end.

5.3.3. Intermediate density penalization

The use of continuous design variables introduces an inherent problem, the appearance of a big amount of regions where the relative density takes intermediate values in the solutions obtained. This circumstance produces amongst other consequences that their manufacturing will have a more elevated expense in case of being possible to manufacture it and at the same time the manufacturing procedure will be also more complex. Consequently, this problem can be mitigated by means of the introduction of factors that penalize the intermediate values of the relative density, diminishing the area where the relative density takes intermediate values.

In this thesis, the method that will be used for penalizing intermediate densities, has been proposed by [París, 2007]. As previously mentioned in chapter 4, this method consists on introducing the penalization factor in the objective function. And for this reason, it will be necessary to modify the objective function that has been formulated in (5.4) in order to consider this penalization effect. This penalization will be incorporated in the same way that in [París, 2007], through a potential function as:

$$F = \sum_{e=1}^{N_e} \int_{\Omega_e} (\rho)^{1/p} d\Omega_e \tag{5.11}$$

where p is the intermediate density penalization factor. The effect of the penalization over the results that will be obtained will be different for different values of p (see figure 5.1).

In the first place, if $p > 1$, an intermediate density penalization over the objective function is introduced, what is the purpose of this procedure. In the second place, if $p < 1$, the opposite effect is achieved, and intermediate density tend to appear in the solution of the problem, what supposes an undesirable effect. Finally, when $p = 1$, no penalization in the objective function is applied.

Furthermore, when the optimal solution has a binary material distribution, there will be an equivalence between minimum cost function and minimum weight function, because penalization of density concerns overall intermediate relative density areas.

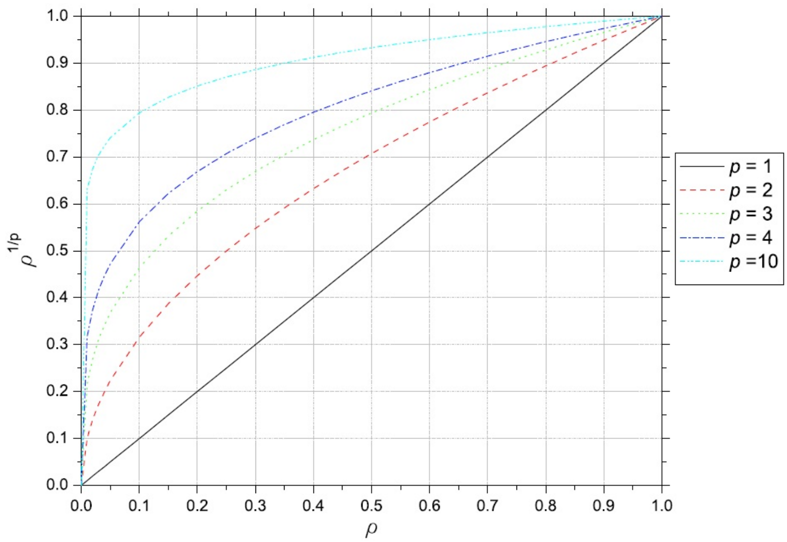


Figure 5.1. Influence of penalty factor over density value [París, 2007]

Then the incorporation of the penalization parameter in the both approaches considered in the previous section will be analyzed.

On the one hand, the incorporation of the penalization parameter in the objective function in the uniform relative density per element approach can be obtained as:

$$F = \sum_{e=1}^{N_e} (\rho_e)^{1/p} \int_{\Omega_e} d\Omega_e = \sum_{e=1}^{N_e} (\rho_e)^{1/p} V_e = \sum_{e=1}^{N_e} W_e \tag{5.12}$$

and similarly to the case without penalization, if all elements of the mesh have the same volume, (5.12) can be reduced to:

$$F = \sum_{e=1}^{N_e} (\rho_e)^{1/p} \tag{5.13}$$

On the other hand, the incorporation of the penalization parameter in the objective function in the material density distributions by means of isogeometric interpolation approach requires to introduce the relative density in the integrals, as in this case density does not take a constant value in each knot span. Due to this, two strategies have been analyzed with the aim of simplifying the procedure, without solving complex integrals.

In the first approach, the original objective function will be integrated first in all the knot spans, and the contribution of each design variable in the whole domain will be combined. Finally, the penalization factor will be applied over the total contribution of each design variable. Furthermore, in this method, the equation of the penalized objective function will be:

$$\begin{aligned} &\text{2D} \qquad F = \sum_{i=1}^{n}\sum_{j=1}^{m} (\rho_{i,j})^{(1/p)} \sum_{e=1}^{N_e} C_{ij,e} V_{ks} n_x n_y \\ &\text{3D} \qquad F = \sum_{i=1}^{n}\sum_{j=1}^{m}\sum_{k=1}^{l} (\rho_{i,j,k})^{(1/p)} \sum_{e=1}^{N_e} C_{ijk,e} V_{ks} n_x n_y n_z \end{aligned} \tag{5.14}$$

where V_{patch} is the volume of the patch, and the coefficient $C_{ijk,e}$ can be obtained as follows:

$$\begin{aligned} &\text{2D} \qquad C_{ij,e} = \int_{\Omega_e} N_{i,p}(\xi) M_{j,q}(\eta) d\xi d\eta \\ &\text{3D} \qquad C_{ijk,e} = \int_{\Omega_e} N_{i,p}(\xi) M_{j,q}(\eta) L_{k,r}(\zeta) d\xi d\eta d\zeta \end{aligned} \tag{5.15}$$

In the second strategy approached, the original objective function will be integrated in each knot span separately, and the penalization factor will be applied over the result of the integral in each knot span. Consequently, the equation of the penalized objective function with this method will be:

$$F = \sum_{e=1}^{N_e} W_e = \sum_{e=1}^{N_e} \rho_e^{(1/p)} V_{ks} \tag{5.16}$$

where ρ_e is the result of the integral of the relative density in the knot span e. This integral can be calculated as:

$$
\begin{aligned}
&\text{2D} \qquad \rho_e = \sum_{i=1}^{n}\sum_{j=1}^{m} \rho_{i,j} n_x n_y \int_{\Omega_e} N_{i,p}(\xi) M_{j,q}(\eta)\ d\xi d\eta \\
&\text{3D} \qquad \rho_e = \sum_{i=1}^{n}\sum_{j=1}^{m}\sum_{k=1}^{l} \rho_{i,j,k} n_x n_y n_z \int_{\Omega_e} N_{i,p}(\xi) M_{j,q}(\eta) L_{k,r}(\zeta)\ d\xi d\eta d\zeta
\end{aligned}
\tag{5.17}
$$

Once both methods have been tested and the results obtained with each of them have been analyzed, the second one is finally chosen. This choice is due to the fact that in the first method design variables are really penalized, that only avoids the existence of intermediate values of design variables. That does not mean that the existence of intermediate relative density values in the domain is avoided. However, the effect that has been pretended with the penalization has been obtained with the second method, where intermediate relative densities are penalized in the whole domain.

5.4. Damage constraint

Once the objective function has been established in both methods, the Finite Element Method and the Isogeometric Analysis, the other aspect that is required in the formulation of the topology optimization problem is the definition of constraints. These constraints will require the calculation of the structural stresses, since the approach developed in this thesis is the minimum weight with stress constraints. When stress constraints are considered, it is necessary to calculate a big amount of stresses in the domain. The number of stresses should be at least the number of design variables of the problem.

In the first place, the first method to consider stress constraints was the local stress constraints approach, in this method one constraint is formulated for each stress. The main problem of this approach appears in problems where a high spatial definition of the solution is required, since the number of design variables that will be used to define the material layout is considerably high, and consequently, the number of constraints that will be imposed will be also considerably high. As a result of this, the amount of time and computational requirements will be also big enough, that makes the solution of the problem an arduous task.

In the second place, the global stress constraint approach that combines all stresses in one constraint, or block aggregated approach of the stress constraints that combines a certain number of stresses in one constraint, are considered. The main problem of these approaches is that it will be impossible to ensure that all stresses, that have been previously calculated, are less than the maximum stress value allowed.

Finally, damage approach will be the alternative proposed in order to avoid both problems. Damage approach in topology optimization problems with stress constraints was firstly proposed by Verbart [Verbart et al., 2016].

In this method, the general concept is to penalize the presence of local failure in a mechanical body through the damage of the material where local failure occurs.

In other words, a perturbation of only one material property is made depending on the amount of local failure. For this purpose, an alternative model of the original mechanical body is created and one of the properties of the original material will be perturbed in this alternative model.

On the one hand, it will be necessary to assume that the overall performance of the structure will have to be a monotonic function of the local material properties, and on the other hand, it will be important that degraded material will never improve its overall performance. Consequently, the damage model will always have a worse performance or at best equal than the original undamaged model.

Therefore, local constraints violations will be prevented indirectly through the imposition of a single constraint where both models should have the same overall performance.

Furthermore, this constraint prevents from local failure in material areas and also contributes to the overall performance of the structure, that will be the same in the original and in the damaged model.

Moreover, in this thesis a minimum weight design that satisfies all the local stress constraints by imposing the condition of equality in the overall performance, will be proposed. So, the concept of damage in the proposed method works as a mechanism to penalize local constraints violation.

On the other hand, the accuracy in the physical damage process will not have importance, since the aim is to obtain an optimized design without damage. This method is also closely related to the topology optimization methods what are based in stress constraints.

In contrast with [Verbart et al., 2016], where structural compliance is used as a measure of the overall performance, the structural weight will be used in this thesis as an alternative to the compliance. This new approach supposes an improvement with the original one, since there is a relationship between the objective function and the property used to create the damaged model.

Therefore, this approach has the advantage that only one constraint will be considered instead of local stress constraints individually. And finally, the amount of time that will be required to solve the topology optimization problem will be reduced significantly.

5.4.1. Formulation of damage approach

Even though an equivalent problem to the topology optimization problem with stress constraints will have to be solved in this thesis, it will be also necessary to establish that stress criterion will be used to check that stresses do not exceed their maximum allowable value. This issue will be deeply analyzed in the following sections.

As mentioned above, in the damage approach, two different models that describe the same mechanical body will have to be defined. They will be known hereinafter as original model and damaged model.

On the one hand, in the original model (figure 5.2a), $\rho(x)$ indicates the density in each point of the domain. Once the original model has been established, the method that will be necessary to follow in the damage approach will have several steps.

In the first place, a structural analysis for the density distribution in the original model will have to be calculated. Thus, the displacements field associated with the original model is obtained, an equivalent stress criterion $\sigma(x)$ will have to be computed.

Then, stresses in all check points are obtained and it will be possible to define the region with violated stresses, that will be called Ω_σ.

The next step is the degradation of the material in regions where stresses exceed the maximum allowable value Ω_σ. For this purpose, one of the properties of the material will be perturbed. The damaged model can be seen in figure 5.2b.

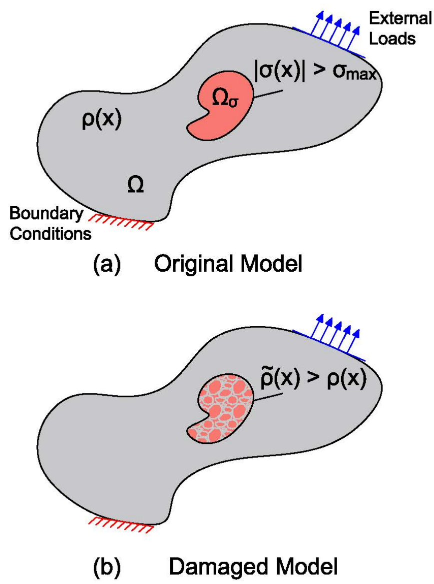


Figure 5.2. Schematic models: (a) the original undamaged model, where the stress exceeds the allowable maximum stress in the red subregion Ω_σ, and (b) the damaged model with degraded material properties in Ω_σ

According to that hereinafter all quantities associated with the damaged model will have a tilde. Finally, the relative density in the damaged model will be defined in the way that the following condition will be satisfied:

$$\begin{cases} \tilde{\rho}(x) > \rho(x), & \forall x \in \Omega_\sigma := \{x \mid \sigma(x) \mid > \sigma_{lim}\}, \\ \tilde{\rho}(x) = \rho(x), & \forall x \in \Omega \backslash \Omega_\sigma \end{cases} \tag{5.18}$$

where $\tilde{\rho}$ has to be strictly positive and higher or equal than the density of the original model (ρ). In the same way that it has been made previously in the original problem, it will be also possible to calculate the displacement fields for the damaged model, but it lacks of interest in this thesis.

Additionally, it is important to take into consideration that the original model will always remain undamaged since the aim of this method is that just the damaged model will be affected by the stress violation.

Furthermore, it will be important that the overall performance of the structure can be measured by a scalar function that only depends monotonically on the local material properties and controls the algorithm.

Consequently, according to (5.18), the damaged model will never have a better performance weight than the original one. Finally, structural weight will be used as a measure of the overall performance in this thesis, since it depends monotonically on the density value. As a result of this, the damaged model will be always more (or at best equally) weighed.

$$\tilde{W} = \int_{\Omega} \tilde{\rho} d\Omega \quad \geq \quad W = \int_{\Omega} \rho d\Omega \tag{5.19}$$

where W and $\tilde{W}$ are the structural weight of the original model and damaged model, respectively. And the relation between both structural weights will be considered to define the inequality constraint that will define the topology optimization problem.

5.4.2. Formulation of the damage constraint

Once the main concepts of Damage Approach have been completely developed, it will be necessary to establish the Damage Constraint that will be used in the topology optimization problem. As mentioned above, the main objective in this thesis, is to find the lightest design without the violation of any local stress constraint.

Thus, according to (5.18) and (5.19), areas where the stresses exceed the maximum allowable stress, will have more weight in the damaged model than in the original one.

Therefore, it will be possible to enforce local stress constraints indirectly through a single inequality constraint, that states that damaged model have to be heavier than original one. As a result of this, damage constraint can be defined as:

$$g(\rho) = \sum_{i=1}^{N_{str}} \left(\frac{\tilde{\rho}_i(\rho, \sigma_i(\rho))}{\rho_i(\rho)} - 1 \right) \leq 0 \tag{5.20}$$

where ρ_i and $\tilde{\rho}_i$ are the value of the relative density in the points where stresses are calculated in the original model and the damaged one, respectively and N_{str} is the number of stresses considered in the formulation of the damage constraint. For this reason, it will be very important to choose the most appropriated points to calculate the structural stresses.

Additionally, g denotes the inequality constraint, what is satisfied as long as the local stress constraints are also satisfied in material regions that contribute to the overall weight. In this case, (5.18) let assure that the modified density can be only greater or equal than the original one, so all fractions in (5.20) will be greater or equal than one.

Finally, when the topology optimization problem (5.3) with the damage constraint (5.20) is numerically solved, it is important to consider that a slightly modification of the damage constraint has to be done, this is due to the way that problem is formulated. If the modification is not done, constraint will be always active or violated, what supposes an important limitation in algorithm operation.

As a consequence of this, a small positive parameter δ will have to be introduced in order to relax the inequality constraint, despite the fact that theoretically, this means that a little failure of stress constraints will be allowed. However, this relaxation, will not suppose a big handicap as it will be analyzed in the next section.

5.4.3. Damage model

In this section, the procedure for doing the material degradation that has been previously introduced in (5.18) will be developed. In the first place, a relationship between density value of the damaged model and the original undamaged model in order to do the material degradation is established as:

$$\tilde{\rho} = \rho_{min} + \beta(\rho - \rho_{min}) \qquad \text{where } \beta(\sigma, \sigma_{lim}) \geq 1 \tag{5.21}$$

Here, ρ_{min} takes the same value that has been previously used to establish an inferior no zero limit of the design variables. This is due to the fact that the singularity of the global stiffness matrix has to be avoided. On the other hand, β is the damage function introduced in order to degrade the material as a function of the ratio between a scalar stress criterion and the maximum allowable stress: $\mid \sigma \mid / \sigma_{lim}$.

Due to simplicity reasons, but without loss of generality, it will be assumed that degradation is based on a single stress value for each point. Additionally, the damage function β will have to be chosen such that (5.21) satisfies (5.18).

Furthermore, two conditions, that damage function have to satisfy, will have to be established in order to have the possibility of solving the problem using gradient-based optimization methods.

On the one hand, damage function should be at least first order differentiable. On the other hand, damage function will have to be monotonically crescent when the stress exceeds its allowable limit.

Despite the fact that there are many functions that satisfy both criteria, an exponential function will be used as a damage function in this thesis. The main reason to use this kind of function is that important penalizations can be obtained for little infringement of the stress constraint. This function will be:

$$\beta(\sigma; \alpha) = \begin{cases} 1, & \text{if } \mid \sigma \mid < \sigma_{max} \\ e^{\alpha(f(\sigma/\sigma_{max}))^2}, & \text{if } \mid \sigma \mid \geq \sigma_{max} \end{cases} \tag{5.22}$$

where $\alpha > 0$ is the exponential degradation parameter that controls the steepness of the damage function. In other words, it controls the amount of damage relative to each stress level. A graphical representation of the damage function for several values of α can be observed in 5.3.

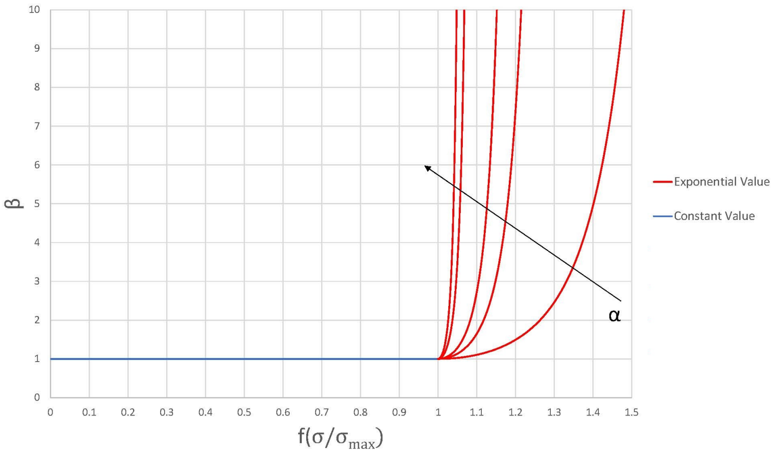


Figure 5.3. Damage function for increasing values of $\alpha > 0$

Moreover, it will be important to take into consideration that it is not intended to model accurately the damage physical process, since the main objective in this thesis is the attainment of an optimal design without damage. In this context, the damage function has an equivalent role to the penalty function to drive the solution towards a design without stress constraints violation.

The main difference between both functions, the damage function and the penalty function, is the part of the optimization problem where each function is introduced. The damage function is the main constraint of the optimization problem, and the penalty function is introduced in the objective function of the problem.

Once theoretical approach of the damage function has been made, the next step is to analyze numerically the damage function. The main purpose of this analysis is to test its behavior in the optimization algorithm.

The main disadvantage of the damage function proposed in (5.22) is that their derivative is 0 when stresses are lower than its maximum allowable value. This supposes an important handicap, when stresses considered are near to their maximum allowable value, since the optimization algorithm used in this situation makes use of the derivative of the damage function. This algorithm will be described in the Chapter 6. Therefore, the damage function (5.22) does not give any proper information to the optimization algorithm.

Consequently, the algorithm would operate meanly, especially in the proximity of the border between feasible region and unfeasible region. As a result of this, an inefficient advance direction will be obtained. Therefore, it will be necessary to introduce some changes in the damage function. This changes will have to improve the operation of the optimization algorithm and they will be described thereupon.

In the first place, the damage function will be translated to the left. As a result of this, the damage function will take a value that will be slightly higher than 1 when stresses will be equal to their maximum value allowable. This does not have a negative effect in the operation of the topology optimization algorithm, because it mitigates the effect of the small positive parameter δ that has to be introduced in order to relax the inequality constraint.

The incorporation of this parameter δ that can be seen in (5.23) supposed that the damaged model can be slightly heavier than the original one. However, the translation of the damage function introduces a little increase of the structural weight in the damaged model when the stresses are lower than their maximum allowable value.

$$g(\rho) = \sum_{i=1}^{N_{str}} \left(\frac{\tilde{\rho}_i(\rho, \sigma_i(\rho))}{\rho_i(\rho)} - 1 \right) \leq \delta \tag{5.23}$$

Despite of this circumstance, this increase in the value of the structural weight when stresses are lower than their maximum allowable value will not be considered, since it will be negligible, if the translation is not big enough. The effect of this translation over the damage function can be seen in (5.24) and in figure 5.4.

$$\beta(\sigma;\alpha) = \begin{cases} 1, & \text{if } \mid \sigma \mid < \sigma_{max}\,(1-\varphi) \\ e^{\alpha(f((\sigma/\sigma_{max}),\varphi))^2}, & \text{if } \mid \sigma \mid \geq \sigma_{max}\,(1-\varphi) \end{cases} \tag{5.24}$$

In the second place, a transition function between the exponential translated damage function (5.24) and the constant value of the damage function will have to be introduced. This transition function will replace the exponential translated damage function (5.24) when stresses are under their maximum value allowable. Moreover, the range where the transition function will be defined will also include a part of the range where the damage function established in (5.24) had a constant value.

The main reason to introduce this transition function is to extend the range where the derivatives of the damage function does not take a null value. Therefore, this transition function will suppose an improvement in the operation of the optimization algorithms. Although the same results can be obtained with a greater translation of the original damage function, this procedure will suppose an important increase in the value of the damage function when stresses are lower than their maximum allowable value due to the use of an exponential function. This circumstance does not allow to consider negligible the damage function when stresses are lower than their maximum allowable value.

This function will have to satisfy several conditions. On the one hand, one of the condition that the original damage function had to satisfy when it was defined was to be at least first order differentiable and monotonically crescent.

For this reason, the transition function will have to maintain the continuity not only of the damage function but also of its first derivative in order to avoid numerical

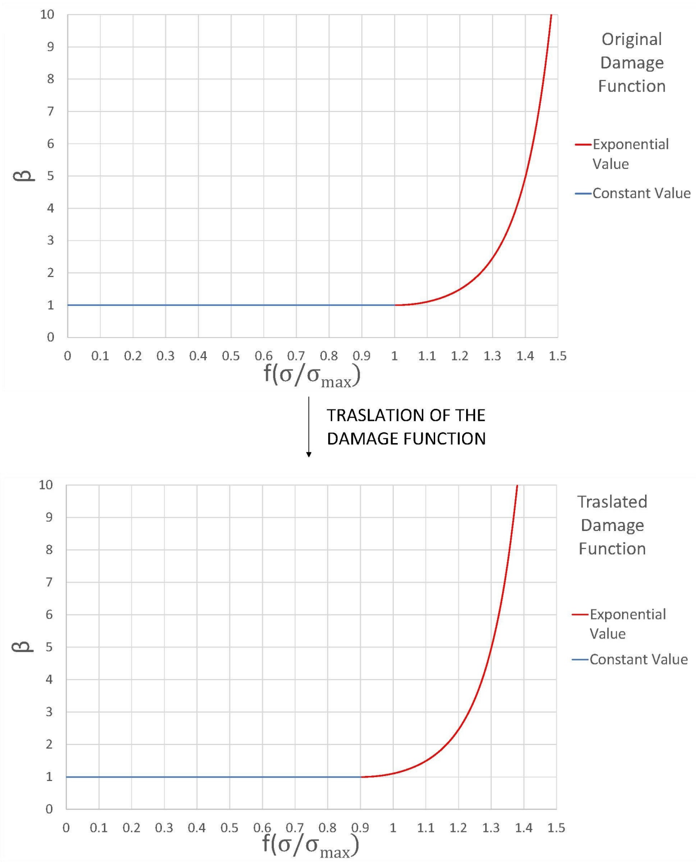


Figure 5.4. Translation of the damage function

problems. Consequently, the value of the transition function and its first derivative will have to coincide with the value of the functions and their first derivatives in the limits of their definition interval. In other words, in the definition of the transition function, these bordering conditions have to be considered.

On the other hand, as it was previously commented, it will be important to take into consideration the fact that the transition function will have to take a definition range bigger than the translation of the damage function, previously made. If this circumstance is not satisfied, the introduction of the transition function would not make sense. This phenomenon can be seen in figure 5.5.

At this point, it is important to establish the basic parameters of both phases: φ is the number of units that the damage function is translated to the left and ε units is the size of the definition range of the transition function. Consequently, $\varphi < \varepsilon$.

Once all the aspects related to the definition of the transition function had been established. Two different approaches will be developed in order to obtain an appro-

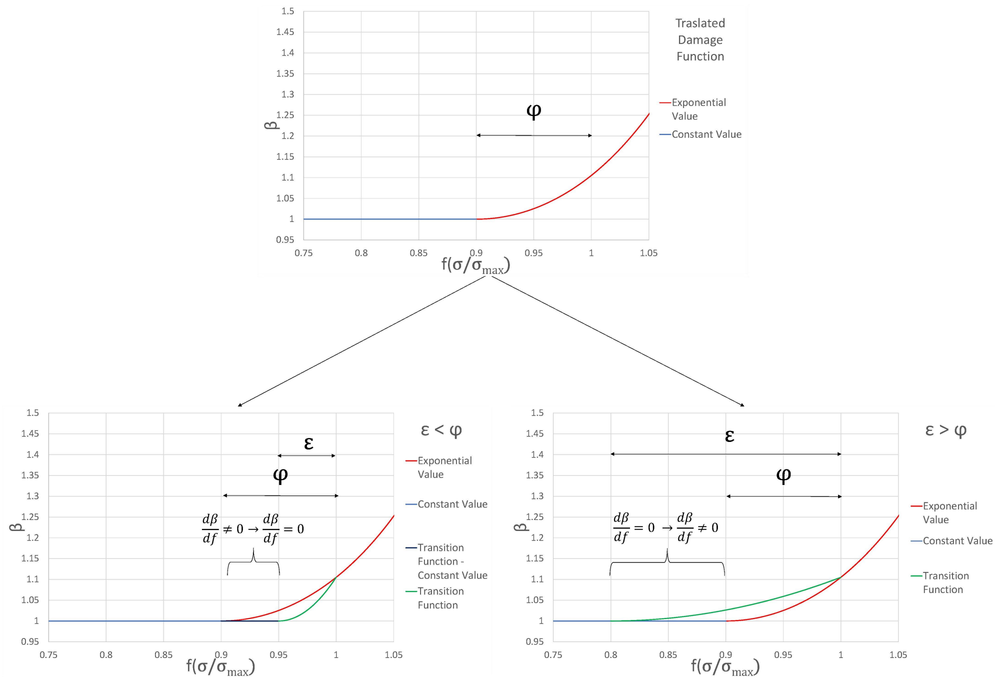


Figure 5.5. Transition function definition

priate definition of the transition function. Prior to that, the general structure of the damage function will be introduced in (5.25).

$$\beta(x) = \begin{cases} 1, & \text{if} \quad x \leq (1-\varepsilon) \\ \text{Transition Function} & \text{if} \quad (1-\varepsilon) < x \leq 1 \\ e^{\alpha(x-1+\varphi)^2} & \text{if} \quad x > 1 \end{cases}$$
$$\beta'(x) = \begin{cases} 0, & \text{if} \quad x \leq (1-\varepsilon) \\ \text{First Derivative of the Transition Function} & \text{if} \quad (1-\varepsilon) < x \leq 1 \\ e^{\alpha(x-1+\varphi)^2} 2\alpha(x-1+\varphi) & \text{if} \quad x > 1 \end{cases} \tag{5.25}$$

where α is a multiplicative coefficient of the exponential function, x will be a function that only depends on the stresses criterion $x = f(\sigma/\sigma_{max})$, φ has been already defined previously, as the number of units that the original damage function has been translated to the left and ε establish the size of the definition range of the transition function.

In the first case, an exponential function will be considered. For this purpose, it will be necessary to modify the values of the coefficients incorporated in the translated exponential function. Therefore, α, φ and the exponent 2 will be replaced by γ, ε and n respectively. Consequently, the problem that is necessary to solve in order to determinate the transition function will be:

$$
\begin{aligned}
\beta(x) &= \begin{cases} 1, & \text{if} \quad x \leq (1-\varepsilon) \\ e^{(\gamma(x-1+\varepsilon))^n} & \text{if} \quad (1-\varepsilon) < x \leq 1 \\ e^{\alpha(x-1+\varphi)^2} & \text{if} \quad x > 1 \end{cases} \\
\beta'(x) &= \begin{cases} 0, & \text{if} \quad x \leq (1-\varepsilon) \\ e^{(\gamma(x-1+\varepsilon))^n} n\gamma^n (x-1+\varepsilon)^{n-1} & \text{if} \quad (1-\varepsilon) < x \leq 1 \\ e^{\alpha(x-1+\varphi)^2} 2\alpha(x-1+\varphi) & \text{if} \quad x > 1 \end{cases}
\end{aligned}
\tag{5.26}
$$

The conditions that will be necessary to impose are continuity of β and β' in the borders of the interval where the transition function is defined $x = (1-\varepsilon)$ and $x = 1$.

In the second case, the use of the translated original exponential function with an auxiliary variable x' in the definition range of the transition function will be analyzed. In this case a change of variable with respect to the original variable x will have to be made $x' = f(x)$. For this purpose, and in the same way that the previous approach, an exponential function will be also proposed. Consequently, the damage function can be formulated as:

$$
\begin{aligned}
\beta(x, x') &= \begin{cases} 1, & \text{if} \quad x \leq (1-\varepsilon) \\ e^{\alpha(x'-1+\varphi)^2} & \text{if} \quad (1-\varphi) < x' \leq 1 \\ e^{\alpha(x-1+\varphi)^2} & \text{if} \quad x > 1 \end{cases} \\
\beta'(x, x') &= \begin{cases} 0, & \text{if} \quad x \leq (1-\varepsilon) \\ e^{\alpha(x'-1+\varphi)^2} 2\alpha(x'-1+\varphi) & \text{if} \quad (1-\varphi) < x' \leq 1 \\ e^{\alpha(x-1+\varphi)^2} 2\alpha(x-1+\varphi) & \text{if} \quad x > 1 \end{cases}
\end{aligned}
\tag{5.27}
$$

Therefore, the function $x' = f(x)$ will have to be defined with the consideration of the next conditions. Damage function and its first derivative will have to be also continuous. However, the introduction of the auxiliary variable maintains the continuity of the damage function. However, it will be necessary to introduce an inherent condition that will have to be satisfied. The value of the limits of the definition range of each interval will have to coincide. For this reason, $f(1-\varepsilon) = 1-\varphi$ and $f(1) = 1$. Moreover, in case of a change of variables, it will be important to take into consideration that:

$$
\frac{d\beta(x')}{dx} = \frac{d\beta(x')}{dx'} \frac{df(x)}{dx} \tag{5.28}
$$

As a result of these considerations, the problem that is necessary to solve in order to determine the transition function will be:

$$
\begin{aligned}
&f(x) = ae^{(\delta(x-1+\varepsilon))^n} \\
&f'(x) = ae^{(\delta(x-1+\varepsilon))^n} n\delta^n (x-1+\varepsilon)^{n-1} \\
&f(1-\varepsilon) = 1-\varphi \qquad\qquad f(1) = 1 \\
&f'(1-\varepsilon) = 0 \qquad\qquad f'(1) = 1
\end{aligned}
\tag{5.29}
$$

Once both problems have been solved for $\varepsilon = 0.1$, $\varphi = 0.01$ and $\alpha = 50$, the transition function in the first case and the change of variable that will have to be introduced in the original exponential function in the second case, will be respectively:

$$
\beta(x) = e^{(\frac{6353}{828}(x-1+0.1))^{20}} \qquad\qquad f(x) = 0.99e^{(\frac{1014}{161}(x-1+0.1))^{\frac{5960}{599}}}
\tag{5.30}
$$

If a comparison between both transition function is made, the maximum difference that has been obtained between both functions is lower than 10^{-5}. For this reason, it can be considered worthless. Finally, the graphic representation of both functions separately and their overlay can be observed in figure 5.6.

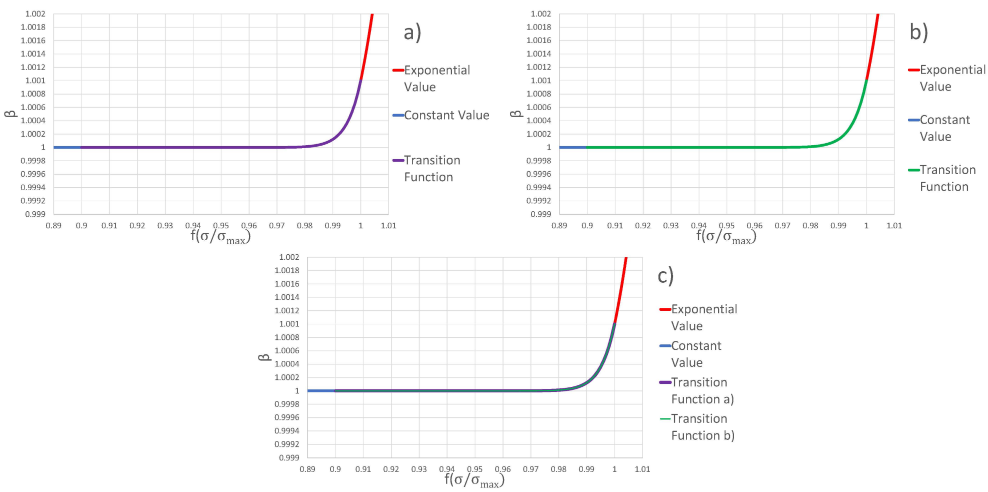


Figure 5.6. Graphic representation: a) damage transition function through an exponential function, b) damage transition function through a variable exchange and c) superposition of both transition functions a) and b), it shows the difference between them

As a result of this analysis, the first approach has been chosen because of their simplicity in comparison with the second one. Thus, the rectified damage function will be:

$$
\beta(x) = \begin{cases} 1, & \text{if} \quad x \leq 0.9 \\ e^{(\frac{6353}{828}(x-0.9))^{20}} & \text{if} \quad 0.9 < x \leq 1 \\ e^{50(x-0.99)^2} & \text{if} \quad x > 1 \end{cases}
\tag{5.31}
$$

where its graphical representation can be seen in figure 5.7. Finally, once damage function has been completely defined, it is necessary to establish the stress criterion to be used. This will be analyzed in the next section.

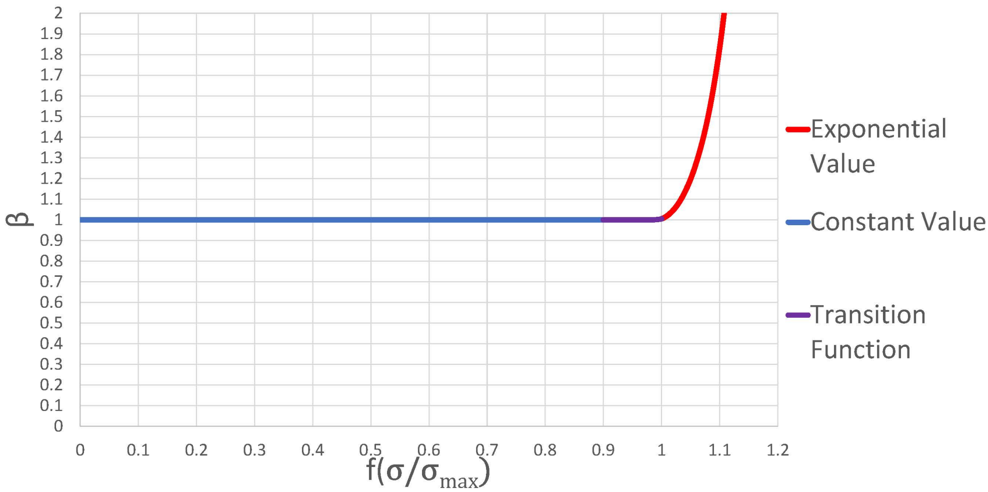


Figure 5.7. Rectified damage function

5.5. Stress criterion

Once damage function has been chosen, the next step to complete the definition of the damage constraint is the choice of the stress criterion. This stress criterion will allow to incorporate the information of the classical stress constraints in the damage constraint. Consequently, the stress criterion chosen will have to be introduced in (5.31) instead of x.

Although the ideal situation will be to establish stress constraint in all the points of the domain, this approach will suppose an important increase of computational requirements and CPU time. Therefore, the most common way to establish stress constraints in the classical approaches of the topology optimization are through the definition of them only in the most representative points of the domain. In other words, the structural stresses will have to be calculated only in a certain number of points of the domain.

In consequence, the next step will be to decide the points where stresses will have to be tested, and to establish the stress criterion that will be used in this thesis to incorporate them to the damage function.

First, the choice of these points is an important aspect, since they have influence on the structural analysis. In the Finite Element Method, they will be located in the central point of each element of the mesh, however, in the Isogeometric Analysis more points have to be used, since the number of design variables used in the Isogeometric Analysis is higher than with the Finite Element Method, for the same meshing.

The same meshing means that the number of elements in the mesh of the Finite Element Method coincides with the number of knot spans in the Isogeometric Analysis. The additional stress points in the Isogeometric Analysis, will have to be near to (or in the same place that) the control points that are not in the central point of any knot span.

Once the points where stresses will have to be calculated are determined the next step is to establish the criterion that will be used to define stress constraints. In general, the stress criterion will be based on material failure criterions, whose formulation is:

$$\begin{aligned} g_{i,1}(\rho) &= \hat{\sigma}(\boldsymbol{\sigma}^h(\mathbf{r}_i^0), \rho) - \hat{\sigma}_{max} \leq 0 \\ g_{i,2}(\rho) &= \hat{\sigma}_{min} - \hat{\sigma}(\boldsymbol{\sigma}^h(\mathbf{r}_i^0), \rho) \leq 0 \end{aligned} \tag{5.32}$$

where $\hat{\sigma}(\boldsymbol{\sigma}^h)$ is the equivalent stress that will be used to define the stress constraint. On the other hand, stress tensor $\boldsymbol{\sigma}$ is established as:

$$\boldsymbol{\sigma} = \left\{ \begin{matrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{matrix} \right\} \tag{5.33}$$

In the case of using steel in the structural design, the most typical breaking criterion is Von Mises, that according to CTE standard [Ministerio de Vivienda, 2006] establish that:

$$\hat{\sigma}_{VM}(\boldsymbol{\sigma}) = \sqrt{\frac{1}{2}\left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)\right]} \tag{5.34}$$

However, CTE standard [Ministerio de Vivienda, 2006] establishes not only that $\hat{\sigma}_{VM} \leq \hat{\sigma}_{max}$, but also that:

$$\sigma^I \leq 2\hat{\sigma}_{max} \quad \text{if} \quad \sigma^I, \sigma^{II}, \sigma^{III} > 0 \tag{5.35}$$

In three-dimensional problems or two-dimensional problems with plane strain, the equation that will have to be used is (5.34). In two-dimensional problems with plane stress, (5.34) can be simplified as:

$$\hat{\sigma}_{VM}(\boldsymbol{\sigma}) = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\tau_{xy}^2} \leq \hat{\sigma}_{max} \tag{5.36}$$

Finally, and according to this approach, stress constraint proposed in (5.32) with equivalent Von Mises stress can be expressed as:

$$g_i(\rho) = \hat{\sigma}_{VM,i}(\boldsymbol{\sigma}^h(\mathbf{r}_i^0), \rho) - \hat{\sigma}_{max} \leq 0 \tag{5.37}$$

where $\hat{\sigma}_{VM,i}(\boldsymbol{\sigma}^h(\mathbf{r}_i^0))$ is the equivalent Von Mises stress that is obtained with structural stress tensor $(\boldsymbol{\sigma}^h(\mathbf{r}_i^0))$, and $\hat{\sigma}_{max}$ is the maximum value allowable of the equivalent stress.

To conclude, the stress criterion that will be introduced in the damage formula of this thesis, is Von Mises criterion. Although, this criterion has been also used in other structural topology optimization papers [Burger & Stainko, 2005], [Duysinx & Bendsøe, 1998], [Duysinx & Sigmund, 1998], [Pereira et al., 2004] and [París, 2007], the main difference consists in the different approach whereby this stress constraint is used in the topology optimization problem. Moreover, in case that other materials will be used in the structural design, its stress criterion can be found in the literature [Duysinx, 1998].

5.5.1. Singularity phenomenon

Once stress criterion that will be used to define stress constraint has been proposed. A numerical analysis of its operation in the algorithm will be developed. This numerical analysis is necessary because there is a problem related with the singularity phenomenon. For this reason, some modifications over the original stress criterion will have to be made.

In the first place, when density tends to zero there are a singularity phenomenon. This is due to the existence of a discontinuity in the stress criterion. This circumstance is really important when for little values of relative density, the stress constraint is violated. This violation disappears when relative density is zero because there is no material in this area. In other words, there are no constraint to impose. This situation can be seen in [Muiños, 2001] [Navarrina et al., 2002], [Navarrina et al., 2005] and [París, 2007].

As a result of this singularity phenomenon, the same approach that had been developed and analyzed in these publications will be used in this thesis. Therefore, the stress criterion considering this singularity phenomenon can be formulated as:

$$g_i(\rho) = \left(\hat{\sigma}_{VM,i}(\boldsymbol{\sigma}_i^h(\rho)) - \hat{\sigma}_{max}\right) \rho_i \leq 0 \tag{5.38}$$

that is generally known as an effective stress constraint.

In the second place, in some cases, the application of effective stress constraint is not sufficient to solve the singularity phenomenon. Therefore, in this case, it is also necessary to introduce an additional relaxation parameter in the formulation [Cheng & Jiang, 1992] and [París, 2007]. Once the additional relaxation parameter has been introduced, the formulation of the stress criterion is:

$$g_i(\rho) = \left(\hat{\sigma}_{VM,i}(\boldsymbol{\sigma}_i^h(\rho)) - \hat{\sigma}_{max}\varphi_i\right) \leq 0 \tag{5.39}$$

where φ_i is the relaxation parameter, that can be calculated as:

$$\varphi_i = 1 - \varepsilon + \frac{\varepsilon}{\rho_i} \tag{5.40}$$

where ε is the relaxation coefficient, and ρ_i is the density in the point where stress is calculated. Relaxation coefficient ε have to take values near to zero ($\varepsilon \in (0.001, 0.1)$) in order to avoid an excessive stress relaxation. The effect of this relaxation coefficient φ_i can be observed in figure 5.8.

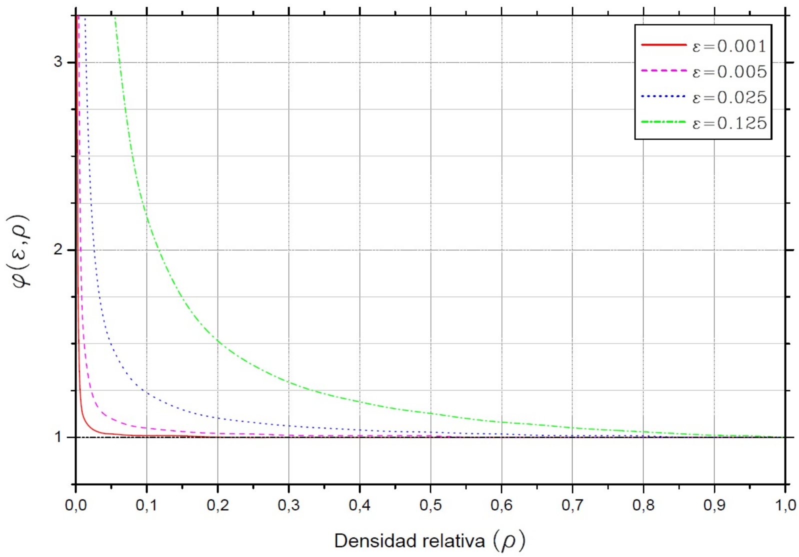


Figure 5.8. Relaxation parameter with different values of ε

Finally, if the relaxation parameters that had been previously described, are incorporated in the general formulation proposed in (5.37). A general approach for stress criterion that will be introduced in damage function is obtained. Its formulation is:

$$g_i(\rho) = \left(\hat{\sigma}_{VM,i}(\boldsymbol{\sigma}_i^h(\rho)) - \hat{\sigma}_{max}\varphi_i\right) \rho_i^q \leq 0 \tag{5.41}$$

where the exponent q is a parameter whose value can be $q = 0$ if real stresses are considered or $q = 1$ in case of effective stresses [París, 2007].

To conclude, an adaptation of the stress criterion (5.41) will be made in order to obtain an expression that depends on $\frac{\hat{\sigma}_{VM,i}}{\hat{\sigma}_{max,i}}$. Therefore, the stress criterion that will be introduced in the damage function is:

$$g_i(\rho) = \left(\frac{\hat{\sigma}_{VM,i}(\boldsymbol{\sigma}_i^h(\rho))}{\hat{\sigma}_{max}\varphi_i} - 1\right) \rho_i^q \leq 0 \tag{5.42}$$

where the term 1 of (5.42) will have to be replaced in the damage function with $1 - \varepsilon$ or $1 - \varphi$ due to the translation that had been done in the damage function and the definition of a transition function in the damage function. Therefore, the damage function obtained in (5.31) will turn into:

$$\beta(x) = \begin{cases} 1, & \text{if} \quad x \leq 0.9 \\ e^{(7.672705\rho^q(x-0.9))^{20}} & \text{if} \quad 0.9 < x \leq 1 \\ e^{50(\rho^q(x-0.99))^2} & \text{if} \quad x > 1 \end{cases} \tag{5.43}$$

where

$$x = \frac{\hat{\sigma}_{VM,i}(\boldsymbol{\sigma}_i^h(\rho))}{\hat{\sigma}_{max}\varphi_i} \tag{5.44}$$

5.6. Overview

In this thesis, the problem of structural topology optimization that has been proposed consists on a minimal weight approach with only one damage constraint where all individual stress constraints are considered together globally instead of having one constraint for each stress individually. For this purpose, in chapter 4 the structural analysis to be used to obtain stress values has been previously developed.

In the first place, the objective function had been formulated. In the minimum weight formulation, a numerical approximation of structural weight is used as an objective function. Moreover, a penalization coefficient is introduced in order to avoid the appearance of intermediate values of density. The formulation of the objective function with this penalization has been developed for both methods: Finite Element Method and Isogeometric Analysis.

In the second place, the damage constraint has been presented and formulated, with a previous description of the damage approach. Then, the damage model has been explained, and finally, the procedure whereby the damage function has been established was explained in detail.

At the end, the stress criterion that will have to be introduced in damage formula has been analyzed. Von Mises stress criterion has been finally proposed in order to be used in this thesis, since steel is the material considered in the design. On the other hand, the minimum number of stresses that should be at least considered in the imposition of the damage constraint has been analyzed, both in the Finite Element Method and in the Isogeometric Analysis. And lastly, singularity phenomenon has been also presented.

The structural topology optimization problem has been completely defined. Thus, in the following chapter, the optimization algorithm developed to solve it will be presented.

To conclude, it is important to consider that hereafter there will be no differences in the approach, neither for the Finite Element Method nor Isogeometric Analysis, and for two-dimensional or three-dimensional problems.

Chapter 6

Optimization algorithms

"Buy only what is necessary, not what is convenient. What is unnecessary, even if it costs one cent, it is expensive."
Lucius Annaeus Seneca, (4BC-65).

6.1. Introduction

Chapter 5 presented the structural topology optimization problem. Now, chapter 6 is devoted to optimization algorithms developed to solve it. The solution procedure can generally suppose an arduous task due to the high number of design variables and also to the non-linearity of the objective function and the damage constraint.

Therefore, algorithms and methods that will be considered in order to solve this kind of problems have to be able to provide an appropriate solution with an assumable quantity of computation requirements.

Moreover, both aspects are equally important because, on the one hand, it can be possible that typical algorithms will not provide an appropriated solution of the problem and, on the other hand, computational requirements can be unacceptable.

The problem that will have to be solved in this thesis presents both challenges. In the first place, a high spatial resolution is required in the solution of the problem. Thus, a high number of design variables will have to be used.

In the second place, objective function can be non-linear if a penalization coefficient of intermediate value of densities is considered. In addition, the damage constraint that has been developed in Chapter 5 is extremely non-linear since it is an exponential function with a high multiplier at the exponent.

As mentioned above, there will be no difference between the use of the Finite Element Method or the Isogeometric Analysis on the one hand and the solution of two-dimensional or three-dimensional problems on the other. This circumstance can be easily explained, since the optimization algorithms only require information about the derivatives of the objective function and the damage constraint. In other words, the way used to define the material layout and the number of dimensions that has the problem formulated does not have any influence over the optimization algorithm chosen to solve the problem.

Since the optimization problem formulated in this thesis is similar to the problem solved in [París, 2007], especially for the case of Global Stress Constraint where there is only one constraint, an evaluation of the advantages and disadvantages of all algorithms that are usually employed in the solution of the Topology Optimization Problem of Structures will not be developed.

As a consequence, the optimization algorithm will be based on the techniques of Sequential Linear Programming (SLP), since Sequential Linear Programming algorithms (SLP) had demonstrated to be efficient in the solution of optimum design problems of structures with constraints [Navarrina, 1987], [Navarrina & Casteleiro, 1991], [Navarrina et al., 2001b], [Navarrina et al., 2001a].

To conclude, this optimization algorithm will be described in this chapter, and crucial aspects that had been commented in previous chapters will be finally justified.

6.2. SLP-LLS algorithm (Sequential Linear Programming with Linear Line Search)

Sequential Linear Programming algorithm (SLP) is based on the advantages of the properties offered by linearized spaces of constraints and the objective function. The solution will be obtained through an iterative method where the behavior of the objective function and the damage constraint will be analyzed by means of the use of their linear approximations. For this purpose, three different approaches will be used.

The most important of them is the Simplex algorithm, that had been developed by [Dantzig & Thapa, 1997], [Dantzig & Thapa, 2003]. This approach can be applied over one linearized space in order to obtain the optimal solution of the topology optimization problem of structures. On the other hand, each iteration of the procedure will consist basically on two steps.

In the first step, the improvement design direction will be obtained through the use of the linear approximations previously commented. This direction will have to satisfy that the largest decrease of the objective function has to be obtained as long as there are not violated constraints.

In the second step, the improvement factor will be calculated in order to modify the original design in the direction that has been previously obtained. This factor will have to ensure that the maximum possible decrease of the objective function is obtained without the violation of any constraint.

Nonetheless, this second step is made at the same time that the first one in case of the Simplex Algorithm, since the Simplex Algorithm takes into consideration the possibility of violating any constraint when the improvement design direction is calculated.

Therefore, the design variables vector will have to be updated in each iteration as:

$$\boldsymbol{\rho}^{k+1} = \boldsymbol{\rho}^k + \theta^k \mathbf{s}^k \tag{6.1}$$

where $\boldsymbol{\rho}^k$ is the design variables vector and θ^k is the improvement factor in the improvement design direction $\mathbf{s}^k$ in the kth iteration.

However, the way improvement design direction $\mathbf{s}^k$ will be calculated will depend on actual state of the design, since there is not any algorithm that can be used in whatever situation. On the other hand, the use of the linear approximation of the problem is the only thing that should be considered for any of the approaches used to calculate the improvement design direction.

In general, the attainment of the improvement design direction is the most important and complex part of the algorithm, since it will require of specific and elaborate techniques due to the high number of design variables considered in the problem.

For all these reasons, linear programming method (Simplex) will be used in this thesis, because it gives acceptable solutions for this kind of problem and it is robust.

Nevertheless, this algorithm will have to be supplemented by others, when there are not active constraints or some constraints are strongly violated, since there is no point in using the Simplex Algorithm if there are not active constraints and the Simplex Algorithm tends to fail when some constraints are strongly violated. In all other cases the Simplex Algorithm is the best approach among the three approaches proposed.

Finally, it is important to remind that a previous analysis of the structural tensional state will have to be done in each iteration, since stresses are the basis in the damage constraint definition. This will have a big influence in the choice of the method that should be used to calculate the improvement design direction.

6.2.1. Steepest descent method

The first situation that will be object of analysis is the case where there are no active constraints. This situation usually appears in the initial iterations of the topology optimization problem, since initial designs are oversized and do not usually have active constraints.

Moreover, Simplex can not be applied in this situation because the only available information is the gradient of the objective function. This situation will continue until one of the constraints of the problem will become active.

Thus, Steepest Descent Method will be the approach that will be used to calculate the improvement design direction in case that there are no active constraints, because it just involves objective function derivatives.

The improvement design direction will coincide with the steepest descent of the objective function in this method. Therefore, the improvement design direction obtained with the Steepest Descent Method will be the opposite to the gradient of the objective function:

$$\mathbf{s}^k = -\frac{dF}{d\boldsymbol{\rho}}\bigg|_{\boldsymbol{\rho}^k} \triangle\boldsymbol{\rho}^k \tag{6.2}$$

where $\mathbf{s}^k$ is the improvement design direction without normalizing in the kth iteration, and $\triangle\boldsymbol{\rho}^k$ is maximum modification of the design variables $\boldsymbol{\rho}$ in each iteration.

However, it is important to consider the fact that the side constraints are not able to introduce any additional information in the method. Because of this, as the optimization algorithm does not take side constraints into consideration, the improvement factor that will be obtained, could be equal to zero, what produces a shutdown of the algorithm.

The easiest way to avoid this problem is through the adjustment of the coefficient $\triangle\boldsymbol{\rho}^k$ in accordance with the minimum distance between each design variable and its side constraint, in each iteration.

At this point it is possible to justify the modification of the original damage constraint developed in Chapter 5. If this modification was not made the Steepest Descent Method will be also used when there are active but no violated constraints. This circumstance is due to the null value of the original damage constraint gradient when the Damage Constraint is active. This situation will persist while the Damage Constraint will not be violated.

Finally, as mentioned above, this algorithm will be generally used in the initial iterations of the topology optimization problem once the first constraint is active, this algorithm will be hardly ever used, because the solution of the topology optimization problem will have a big amount of stresses near to their maximum allowable value. That is, damage constraint will be most certainly active in this situation.

6.2.2. Simplex algorithm

The second situation that will be object of analysis is the case when there are only active constraints. The Simplex algorithm will be the method that will be used in this situation. On the other hand, the use of this method will not be advisable if some constraint is strongly violated. In this situation this algorithm can fail and there are better methods.

Furthermore, the Simplex algorithm will be the most important algorithm to obtain the improvement design direction. This importance is due to the fact that it will be used in a high percentage of the iterations required to find the optimum solution of the Topology Optimization problem.

This circumstance can be explained since the number of stresses that are near to their maximum allowable value will be increased as the design is closer to the optimal design. In other words, the damage constraint will be active during the majority of the procedure.

On the other hand, the Simplex algorithm will be the basis of the topology optimization problem since it operates in the border between feasible and unfeasible region.

This method will provide an improvement design direction that will take all stress constraints, through the damage constraint, and side constraints into consideration.

Topology optimization problems with stress constraints usually involve a very large number of constraints since it is necessary to check the stress state in a very large number of points of the domain. However, this does not suppose a big problem in this thesis because there is only one constraint: The Damage Constraint.

As mentioned above, the operation of this algorithm is based on the application of Linear Programming techniques over a linearized space. Consequently, the first order Taylor Series will have to be developed for the objective function and also for the damage constraint in the proximity of the previous solution $\left(\boldsymbol{\rho}^k\right)$ as:

$$
\begin{aligned}
F(\boldsymbol{\rho}^{k+1}) &\approx F(\boldsymbol{\rho}^k) + \left.\frac{dF}{d\boldsymbol{\rho}}\right|_{\boldsymbol{\rho}^k} \theta^k \mathbf{s}^k \\
g(\boldsymbol{\rho}^{k+1}) &\approx g(\boldsymbol{\rho}^k) + \left.\frac{dg}{d\boldsymbol{\rho}}\right|_{\boldsymbol{\rho}^k} \theta^k \mathbf{s}^k
\end{aligned}
\tag{6.3}
$$

where $\mathbf{s}^k$ is the improvement design direction vector and θ^k is the improvement factor.

Finally, both inferior and superior limits of the design variables will have to be taken into consideration in the application of Simplex Algorithm when side constraints are considered. Side constraints of the linearized problem will also consider the use of moving limits around the previous solution to keep the precision of the first order approach.

As a result of this, Simplex algorithm will be applied over the linearized problem in order to obtain the improvement design direction in each iteration. A full development of the Simplex algorithm can be seen in [París, 2007].

6.2.3. Back to the feasible region

Lastly, the third situation that will be object of analysis is the case when there are constraints that are strongly violated. In this situation, the Linear Programming Algorithm may fail.

Therefore, if linear programming algorithms are used with strongly violated constraints, it is highly possible that a design without strongly violated constraints could not be obtained.

Because of this, it will be necessary to have an alternative algorithm that provides an improvement design direction that ensures the introduction of the solution in the region where constraints are not strongly violated in order to have the possibility of continuing with the solution of the topology optimization problem. Thus, Linear Programming Algorithms would be used in the next iterations.

The algorithm used in this case is known as Back to the feasible region method. This algorithm has a similar basis to the Steepest descent method, with the difference that the strongly violated constraint gradient is used instead of the objective function one. This circumstance is due to the fact that the main objective of this algorithm is to reduce the value of the strongly violated constraint. As a consequence, the equation that will have to be used in order to obtain an appropriate improvement design direction is:

$$\mathbf{s}^k = -g \frac{\left.\frac{dg}{d\boldsymbol{\rho}}\right|_{\boldsymbol{\rho}^k}}{\left\|\left.\frac{dg}{d\boldsymbol{\rho}}\right|_{\boldsymbol{\rho}^k}\right\|^2} \triangle\boldsymbol{\rho}^k \tag{6.4}$$

where $\mathbf{s}^k$ is the non-normalized improvement design direction at the kth iteration, and $\triangle\boldsymbol{\rho}^k$ is the value of the maximum modification of design variables at each iteration.

To conclude, once the improvement design direction is obtained it is necessary to normalize it and to proceed to compute the improvement factor θ^k.

6.2.4. Computation of the advance step

Once the improvement design direction $\mathbf{s}^k$ has been calculated and normalized, the next step is to obtain the most appropriate value of the improvement factor θ^k.

In order to calculate it, a linear line search will be made in the direction that has been previously calculated. The first order Taylor Series will have to be developed for not only the objective function but also for the damage constraint in order to do the linear line search.

The choice of the linear line search has been due to the fact that a big amount of computational resources would be necessary to calculate the second derivative in spite of requiring only the second order directional derivative, particularly when thousands of design variables are involved.

Therefore, it will be only necessary to calculate the first order directional derivatives. Their calculation is not an arduous task since the information required to calculate them has been obtained previously: the first order derivative vector and the improvement design direction vector. Consequently, the first order directional derivative can be obtained by means of the scalar product between both vectors.

Once the first order directional derivatives of the objective function and the damage constraint are obtained, their first order Taylor Series can be calculated as:

$$\begin{aligned} F(\boldsymbol{\rho}^{k+1}) &\approx F(\boldsymbol{\rho}^k) + \left. D_s F\right|_{\boldsymbol{\rho}^k} \theta^k \\ g(\boldsymbol{\rho}^{k+1}) &\approx g(\boldsymbol{\rho}^k) + \left. D_s g\right|_{\boldsymbol{\rho}^k} \theta^k \end{aligned} \tag{6.5}$$

where $\left. D_s F\right|_{\boldsymbol{\rho}^k}$ is the first order directional derivative of the objective function, $\left. D_s g\right|_{\boldsymbol{\rho}^k}$ is the first order derivative of the Damage Constraint and θ^k is the advance factor.

Once the first order Taylor Series have been established, the advance factor can be calculated. First, the minimum and the maximum values of the advance factor will have to be established, provided that side constraints of design variables can not be violated.

Then, the algorithm calculates the value of the objective function and the damage constraint by using the first order Taylor Series, in a set of close and equally spaced points in the range of admissible values of the advance factor.

Finally, the most adequate value of the advance factor will be computed. The advance factor will make the objective function minimum without violating the damage constraint.

This value will generally coincide with the maximum possible advance factor without the violation of damage constraint, since first order Taylor Series are used, as in SPL. On the other hand, it is important to emphasize that this step would not be necessary if the normalization of the improvement design direction vector has not been made in case of the Simplex Algorithm, since Simplex Algorithm provides directly the most adequate modification of the design variables in each iteration.

Moreover, this algorithm has also demonstrated to be efficient in spite of being necessary to calculate several derivatives and make the calculation in a certain set of points. Despite of this, a more accurate choice of the advance factor can be made, since there are only first degree equations in the algorithm. In addition, it will be only necessary to control the value of one constraint and the side constraints of the design variables.

6.3. Side constraints

Although optimization algorithms had been completely developed, one component of the topology optimization problem has not been analyzed in depth. For this reason, a study of the side constraints of the design variables will be made in this section.

In spite of having considered only the state of the damage constraint in the choice of the optimization algorithm, the influence of side constraints in the optimization algorithms will have to be considered.

In addition, it will be important to remember that the damage constraint is extremely non-linear, that requires efficient optimization algorithms. On the contrary, side constraints of design variables are completely linear; and as a consequence, they will be easier to implement than the damage constraint.

On the other hand, if the side constraints are not taken into consideration in the calculation of the improvement design direction, the optimization algorithm will make wrong decisions. In other words, the improvement design direction obtained in this case will be wrong, and as a consequence the advance factor could be lower or equal to zero. Therefore, the improvement design direction will have to take into consideration the side constraints in order to operate properly.

As mentioned above, the improvement of the solution is made in two steps. First, the best improvement design direction will be obtained. Then, in the second step, the magnitude of the advance factor will be determined.

Moreover, it makes no sense to exclude side constraints from the computation of the improvement design direction, especially when a full-void design is intended through the introduction of an intermediate density penalization. As a consequence, all design variables will tend to take values near to their side constraints.

Notwithstanding, it is important to remark that the main objective of this thesis has been to develop an adequate model with only one constraint, that will include efficiently the effects of a higher set of local stresses previously calculated. In addition, side constraints will have to be also considered since in other case extremely counterproductive effects will be obtained.

The Simplex algorithm takes into consideration the side constraints in the calculation of the improvement design direction for itself. The Steepest Descent Method and the Back to the Feasible Region method require the introduction of a verification in order to take into consideration the side constraints. This verification changes to zero the value of the components of improvement design direction vector, whose design variables take a value equal to its side constraint, and the improvement design direction will produce its exit from its range of admissible values.

To conclude, a deep analysis of the behavior of side constraints in topology optimization problems, and the reasons why they have to be considered in the calculation of the improvement design direction had been made in [París, 2007].

6.4. Overview

The topology optimization problem of structures requires specific techniques to solve it due to its particular characteristics and requirements.

Although, the topology optimization problem that has been developed in this thesis requires less computational requirements than the similar problems that had been previously formulated. Despite of this, some particular characteristics such as the high non-linearity of the damage constraint makes necessary to use specific algorithms.

Moreover, the use of only one constraint to take into account the stress state of the structure, makes possible the solution of topology optimization problems of structures with a high number of design variables, that means the attainment of solutions with high spatial definition in the domain.

As mentioned above, the damage constraint is highly non-linear. Nevertheless, if penalization of intermediate values of density is introduced in the formulation, the objective function will be also non-linear. And this fact will mean an additional difficulty to solve the topology optimization problem.

The Sequential Linear Programming algorithm has been chosen to solve the topology optimization problem. This algorithm includes 3 different approaches regarding the actual constraints state: steepest descent, simplex and feasible direction. Additionally, this algorithm has been complemented with a Linear Line Search.

Even though, the Simplex algorithm is the most used approach in the solution of the topology optimization problem. This algorithm needs to be complemented with 2 gradient methods, when there are no active constraints and if there are any strongly violated constraints. In both situations the improvement design direction will be the opposite of the gradient of the objective function and the damage constraint respectively.

Furthermore, a brief analysis about the importance of considering the side constraints of the design variables has been made. As a result of this analysis, side conditions will have to be considered when the improvement design direction is calculated. Otherwise the advance factor can be easily zero and the optimization algorithm stucks.

Both side constraints, inferior and superior, will have to be considered, because in this thesis full-void designs will be intended through the penalization of intermediate values of the density. That supposes that side constraints will tend to be active.

To conclude, in next chapter, the first order sensitivity analysis will be developed, not only for the objective function, but also for the damage constraint. Moreover, the techniques and procedures used to obtain the information required in the optimization algorithms previously described, will be deeply analyzed and developed.

CHAPTER 7

Sensitivity analysis

"Yesterday I was clever, so I wanted to change the world. Today, I am wise, so I am changing myself"
Yalal ad-Din Muhammad Rumi, (1207-1273).

7.1. Introduction

The topology optimization problem of structures with high spatial definition that will be solved in this thesis, require a large number of design variables. As a result of this, the sensitivity analysis will suppose a high percentage of the CPU time that will be needed in order to solve the topology optimization problem. However, the memory computational requirements will also be important because of not only the number of design variables but also the way used to define the material layout in the domain.

The topology optimization problem of structures proposed in this thesis has been formulated previously in Chapter 5 as:

$$\begin{array}{lll} \text{Calculate} & \boldsymbol{\rho} = \{\rho_i\} & i = 1, ..., n \\ \text{ that minimizes} & F(\boldsymbol{\rho}) & \\ \text{ verifying} & g(\boldsymbol{\rho}) \leq 0 & \\ & \rho_{min} \leq \rho_i \leq \rho_{max} & i = 1, ..., n \end{array} \tag{7.1}$$

where $\boldsymbol{\rho}$ is the design variables vector, F is the objective function, g is the damage constraint and ρ_{min} and ρ_{max} are the side constraints of the design variables.

Furthermore, the methods that will have to be used in the solution of the topology optimization problem (7.1) have been developed in Chapter 6.

First of all, the derivatives of the objective function and damage constraint will be calculated according to the requirements of these techniques. The optimization methodology developed in Chapter 6, not only for the Simplex Algorithm but also for the Steepest Descent Method, looks for the optimum solution iteratively as:

$$\boldsymbol{\rho}^{k+1} = \boldsymbol{\rho}^{k} + \theta^{k}\mathbf{s}^{k} \tag{7.2}$$

where $\mathbf{s}^k$ is the improvement design direction obtained through the Sequential Linear Programming, and θ^k is the advance factor that has been obtained once the improvement design direction has been calculated, through a linear line search in the direction $\mathbf{s}^k$.

Consequently, the method proposed requires at least the calculation of the complete first order sensitivity analysis of the objective function and the damage constraint, in order to calculate the improvement design direction. Although the calculation of the advance factor could be made by considering a high order sensitivity analysis [Navarrina et al., 2000], this process has been discarded due to computational issues.

Although in the last sentence of Chapter 5, it was said that there will not be more differences in the approach that is being analyzed with respect to the use of both formulations: the Finite Element Method or the Isogeometric Analysis. The influence that each design variable has over each element or knot span of the mesh will suppose a little difference in the formulation of the sensitivity analysis.

Whereas this influence will be unequivocal between design variable and element in the case of the most usual Finite Element Models, in the Isogeometric Analysis proposed, each design variable will have influence over a set of knot spans. This issue will be deeply analyzed in next section.

Finally, in this chapter, the complete first order sensitivity analysis of the objective function and the damage constraint will be developed.

7.2. Preliminary concepts

In the first place, notational concepts that will have to be used during the development of the sensitivity analysis will be established in order to avoid having to explain them every time when they will be used.

First of all, the design variables vector of the topology optimization problem can be defined as:

$$\boldsymbol{\rho} = \left\{ \begin{matrix} \rho_1 \\ \vdots \\ \rho_n \end{matrix} \right\} \tag{7.3}$$

Moreover, a function that depends on the design variables can be expressed as:

$$f(\boldsymbol{\rho}) = f(\rho_1, \rho_2, ..., \rho_n). \tag{7.4}$$

Furthermore, its first derivative can be expressed as:

$$\frac{df}{d\boldsymbol{\rho}} = \nabla f = \begin{Bmatrix} \frac{\partial f}{\partial \rho_1} & \frac{\partial f}{\partial \rho_2} & \cdots & \frac{\partial f}{\partial \rho_n} \end{Bmatrix} \tag{7.5}$$

Finally, its first directional derivative can be calculated as:

$$D_s f = \frac{df}{d\boldsymbol{\rho}} \frac{d\boldsymbol{\rho}}{d\theta} = \frac{df}{d\boldsymbol{\rho}} \mathbf{s} \tag{7.6}$$

where $\mathbf{s}$ is the improvement design direction, that has been obtained with the Sequential Linear Programming.

7.2.1. Density considerations

First of all, and as mentioned above, there will be some important differences between the Finite Element Method and the Isogeometric Analysis proposed in this thesis to solve the topology optimization problem. This differences are related with the way to define the relative density in the structural domain, and as a result, they have to be considered in the calculation of the sensitivity analysis.

Therefore, these differences will be analyzed in this section, in order to avoid being repetitive in the rest of the chapter.

It will be necessary to use two different kinds of density in the definition of the objective function and the damage constraint. These two densities are: the elemental density that will be used to define the objective function and the local density that will be necessary in the definition of the damage constraint. These concepts will be explained hereunder.

On the one hand, the objective function has been defined by means of the elemental density ρ_e. This elemental density is equal to the mean value of the relative density in the element considered, and consequently, the value of the elemental density will be equal in both cases, with the original relative density distribution and with the elemental density.

This way to define the objective function has as an objective to avoid the appearance of undesirable phenomena when the intermediate values of the relative density are penalized. In the Finite Element Method proposed the relation between the relative density (design variables) and the elemental density is direct, however, in the Isogeometric Analysis this connection is more complicated. Thus, the elemental density in case of the Finite Element Method proposed can be defined as:

$$(\rho_e)_{FEM} = \rho_i \tag{7.7}$$

where ρ_i is the value of the density in the element e, since each design variable defines the relative density in one element of the mesh.

On the contrary, the elemental density in case of the Isogeometric Analysis requires the calculation of all the shape function integrals in the elemental domain. Therefore, the elemental density can be calculated as:

$$
\begin{array}{ll}
\text{2D} & (\rho_e)_{IGA} = \displaystyle\sum_{i_x=1}^{n_x}\sum_{i_y=1}^{n_y} \rho_i n_x n_y \int_{\Omega_e} N_{i_x,p}(\xi) M_{i_y,q}(\eta)\ d\xi d\eta \\
\text{3D} & (\rho_e)_{IGA} = \displaystyle\sum_{i_x=1}^{n_x}\sum_{i_y=1}^{n_y}\sum_{i_z=1}^{n_z} \rho_i n_x n_y n_z \int_{\Omega_e} N_{i_x,p}(\xi) M_{i_y,q}(\eta) L_{i_z,r}(\zeta)\ d\xi d\eta d\zeta
\end{array}
\tag{7.8}
$$

where ρ_i is the value of the relative density in the control point i and $N_{i_x,p}$, $M_{i_y,q}$ and $L_{i_z,r}$ are the B-splines that defines the influence of design variable i over the domain. The index i makes reference to the global numeration of the design variables, and the indexes i_x, i_y, i_z makes reference to the matrix structure that has been used in the definition of the B-splines surfaces or B-splines solids in Chapter 3.

Once the elemental density has been defined, the first order derivative of the elemental density with respect to the design variables is:

$$
\left(\frac{d\rho_e}{d\rho_i}\right)_{FEM} = \begin{cases} 1 & \text{if the element } e \text{ is defined with } \rho_i \\ 0 & \text{if the element } e \text{ is not defined with } \rho_i \end{cases}
\tag{7.9}
$$

$$
\begin{array}{ll}
\text{2D} & \left(\dfrac{d\rho_e}{d\rho_i}\right)_{IGA} = n_x n_y \displaystyle\int_{\Omega_e} N_{i_x,p}(\xi) M_{i_y,q}(\eta)\ d\xi d\eta \\
\text{3D} & \left(\dfrac{d\rho_e}{d\rho_i}\right)_{IGA} = n_x n_y n_z \displaystyle\int_{\Omega_e} N_{i_x,p}(\xi) M_{i_y,q}(\eta) L_{i_z,r}(\zeta)\ d\xi d\eta d\zeta
\end{array}
\tag{7.10}
$$

On the other hand, the damage constraint will be defined via local density ρ_s, where ρ_s is the value of the relative density in the point where the stresses had been calculated. In consequence, the local density can be defined in case of the Finite Element Method as:

$$
(\rho_s)_{FEM} = \rho_i \tag{7.11}
$$

where ρ_i is the value of the relative density in element e, that contains the point s where stress is calculated.

In the same way, the local density in case of the Isogeometric Analysis can be defined as:

$$
\begin{array}{ll}
\text{2D} & (\rho_s)_{IGA} = \displaystyle\sum_{i_x=1}^{n_x}\sum_{i_y=1}^{n_y} \rho_i N_{i_x,p}(\xi) M_{i_y,q}(\eta) \\
\text{3D} & (\rho_s)_{IGA} = \displaystyle\sum_{i_x=1}^{n_x}\sum_{i_y=1}^{n_y}\sum_{i_z=1}^{n_z} \rho_i N_{i_x,p}(\xi) M_{i_y,q}(\eta) L_{i_z,r}(\zeta)
\end{array}
\tag{7.12}
$$

where ρ_i is the value of the relative density in each control point and $N_{i_x,p}$, $M_{i_y,q}$ and $L_{i_z,r}$ are the value of each B-spline that defines the influence of each design variable in the point where stresses have been calculated. Thus the control points of the B-splines mesh define the relative density distribution and ”s” is a point where stresses are computed.

In a similar way that for elemental density, the first order derivative of the local density at any point with respect to the design variables in case of the Finite Element Method is:

$$\left(\frac{d\rho_s}{d\rho_i}\right)_{FEM} = \begin{cases} 1 & \text{if the stress is located in the element defined with } \rho_i \\ 0 & \text{if the stress is not located in the element defined with } \rho_i \end{cases} \tag{7.13}$$

In case of the Isogeometric Analysis the first order derivative of the local density at any point with respect to the design variables can be calculated as:

$$\begin{aligned} &\text{2D} \qquad \left(\frac{d\rho_s}{d\rho_i}\right)_{IGA} = N_{i_x,p}(\xi)M_{i_y,q}(\eta) \\ &\text{3D} \qquad \left(\frac{d\rho_s}{d\rho_i}\right)_{IGA} = N_{i_x,p}(\xi)M_{i_y,q}(\eta)L_{i_z,r}(\zeta) \end{aligned} \tag{7.14}$$

7.3. Derivative methods

Once the notational concepts have been established and density considerations have been analyzed, the next step is the development of the differentiation method to be used in the attainment of the value of the derivatives for the objective function and also the damage constraint.

In the first place, it will be necessary to analyze the way objective function and the damage constraint are defined as function of the design variables before developing the sensitivity analysis.

Moreover, it will be possible in this case to establish a difference between functions that depends directly on the design variables, such as the structural weight (the objective function), and functions that depends on the design variables through intermediate variables, such as the structural displacements or the structural stresses that are used in the definition of the damage constraint.

The derivatives of functions that depends directly on the design variables can be calculated analytically through the use of the chain rule. However, if they do not depend directly the procedure whereby the derivatives of this functions will be obtained is more complex.

Therefore, the method that will be used to calculate these derivatives will be the adjoint variable approach, that has been previously applied and analyzed in [Lee, 1999] [Cao et al., 2002], [París, 2007], [Chung et al., 2009].

Other procedures have been also considered and analyzed, but they have been discarded, because of the big amount of structural analysis per iteration in the case of numerical differentiation and the high number of systems of linear equations that will have to be solved in the case of direct analytical differentiation.

To conclude, these problems becomes more and more expensive as the number of design variables increases, what happens in case of the topology optimization problem in that a high spatial definition will be intended.

7.3.1. Analytical Differentiation through the Adjoint Variable Approach

The direct analytical differentiation and the analytical differentiation through the adjoint variable approach are two different analytical methods to deal with sensitivity analysis. The choice of the most suitable method will depend on the relationship between the number of design variables and the number of constraints.

On the one hand, the direct analytical differentiation is advisable when the number of design variables is lower than the number of constraints, since the number of systems of linear equations that will have to be solved is equal to the number of design variables. On the other hand, the analytical differentiation through the adjoint variable approach is advisable if the number of constraints is lower to the number of design variables, in this situation it will be necessary to solve only as systems of linear equations as the number of constraints.

As a result of this, the most appropriated method to do the sensitivity analysis in the problem of minimum weight with the Damage Constraint will be the analytical differentiation through the adjoint variable, since only one system of linear equations will have to be solved. Moreover, it will not be necessary to store the displacements first order derivative vector since it will be used to solve only one system of linear equations.

The damage constraint can be formulated as:

$$g(\boldsymbol{\rho}) = g(\boldsymbol{\sigma}(\boldsymbol{\alpha}), \boldsymbol{\rho})\big|_{\boldsymbol{\alpha}(\boldsymbol{\rho})} \tag{7.15}$$

where $\boldsymbol{\rho}$ is the design variables vector, $\boldsymbol{\sigma}(\boldsymbol{\alpha})$ are the structural stresses obtained through the structural displacements, and $\boldsymbol{\alpha}$ is the nodal structural displacements vector that has been obtained in the structural analysis.

Therefore, the first order derivative of the damage constraint can be calculated as:

$$\frac{dg}{d\boldsymbol{\rho}} = \left.\frac{\partial g}{\partial \boldsymbol{\sigma}}\right|_{\substack{\boldsymbol{\sigma}(\boldsymbol{\alpha}) \\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\boldsymbol{\sigma}}{d\boldsymbol{\alpha}}\right|_{\boldsymbol{\alpha}(\boldsymbol{\rho})} \frac{d\boldsymbol{\alpha}}{d\boldsymbol{\rho}} + \left.\frac{\partial g}{\partial \boldsymbol{\rho}}\right|_{\substack{\boldsymbol{\sigma}(\boldsymbol{\alpha}) \\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \tag{7.16}$$

where the first order derivatives of nodal displacements can be obtained through the use of the state equation, that has been introduced in (4.18). This state equation can be formulated, both for the Finite Element Method and the Isogeometric Analysis, as a system of linear equations:

$$\mathbf{K}\boldsymbol{\alpha} = \mathbf{f} \tag{7.17}$$

with

$$\mathbf{K} = \mathbf{K}(\boldsymbol{\rho}) \qquad \boldsymbol{\alpha} = \boldsymbol{\alpha}(\boldsymbol{\rho}) \qquad \mathbf{f} = \mathbf{f}(\boldsymbol{\rho}) \tag{7.18}$$

where $\mathbf{K}$, $\boldsymbol{\alpha}$ and $\mathbf{f}$ are the stiffness matrix, the nodal displacement vector and the nodal loads vector respectively.

The first order derivative of the damage constraint requires the calculation of the derivative of the nodal displacement vector with respect to all the design variables. This circumstance supposes the solution of as structural analysis as the number of design variables, with a little modification of only one design variable with respect to the structural analysis calculated previously.

However, it is possible to avoid this arduous task in terms of computational requirements and CPU time, if the state equation (7.18) is derived with respect to each design variable, and the terms obtained are reorganized.

$$\mathbf{K}\frac{d\boldsymbol{\alpha}}{d\boldsymbol{\rho}} = \frac{d\mathbf{f}}{d\boldsymbol{\rho}} - \frac{d\mathbf{K}}{d\boldsymbol{\rho}}\boldsymbol{\alpha} \tag{7.19}$$

In this system of linear equations, the independent term can be calculated through the formulation of the structural analysis developed in (4.20).

The first factor of the independent term is the first order derivative of the nodal load vector with respect to the design variables. Moreover, it is important to take the dependence of the nodal load vector with respect to the design variables by means of the structural weight into consideration. As a result of this, this term can be obtained as:

$$\frac{d\mathbf{f}_k}{d\rho_e} = \iiint_{E_e} ({\boldsymbol{\Phi}_{\boldsymbol{k}}}^T\mathbf{b} - (\mathbf{L}\boldsymbol{\Phi}_{\boldsymbol{k}})^T\mathbf{D}(\mathbf{L}\mathbf{u}^p))d\Omega \tag{7.20}$$

where

$$\mathbf{f} = \{f_k\}_{k=1,\ldots,N} \tag{7.21}$$

In both equations, N is the number of nodes of the mesh, $\mathbf{f}_k$ is the nodal load vector that is applied over the node k, $\mathbf{u}^p$ is the prescribed nodal displacement vector of the structure, and finally, ρ_e is the elemental density value in each element of the mesh.

The second factor of the independent term that corresponds to the first order derivative of the stiffness matrix multiplied by the nodal displacements vector can be easily calculated.

Furthermore, the first order derivative of the stiffness matrix can be calculated in a similar way that the first order derivative of the nodal load vector as:

$$\frac{d\mathbf{K}_{ji}}{d\rho_e} = \iiint_{\Omega_e} (\mathbf{L}\boldsymbol{\Phi}_j)^T\mathbf{D}(\mathbf{L}\boldsymbol{\Phi}_i)d\Omega \tag{7.22}$$

where

$$\mathbf{K}_{ji}(\boldsymbol{\rho}) = \sum_{e=1}^{N_e} \mathbf{K}_{ji}^{e}(\boldsymbol{\rho}) \tag{7.23}$$

At this point it is important to see the fact that equations (7.20) and (7.22) are equivalent to the equations (4.20), if the density takes the unitary value at every point of the domain, or in other words, all design variables will be equal to 1.

Because of this, the sensitivity analysis of nodal loads vector can be obtained by means of the volumetric forces applied over the structure and the prescribed nodal displacements, since in this case external loads will never depend on the design variables, and as a consequence, their derivatives will always be zero.

On the other hand, the sensitivity analysis of the stiffness matrix can be obtained in the same way that the elemental stiffness matrix with the previous considerations.

Finally, the derivative of the nodal displacements vector can be obtained as:

$$\frac{d\boldsymbol{\alpha}}{d\boldsymbol{\rho}} = \mathbf{K}^{-1}\left(\frac{d\mathbf{f}}{d\boldsymbol{\rho}} - \frac{d\mathbf{K}}{d\boldsymbol{\rho}}\boldsymbol{\alpha}\right) \tag{7.24}$$

Additionally, the derivative of the nodal displacement vector (7.24) is replaced in the expression of the first order derivative of damage constraint as:

$$\frac{dg}{d\boldsymbol{\rho}} = \left.\frac{\partial g}{\partial \boldsymbol{\sigma}}\right|_{\substack{\boldsymbol{\sigma}(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\boldsymbol{\sigma}}{d\boldsymbol{\alpha}}\right|_{\boldsymbol{\alpha}(\boldsymbol{\rho})} \mathbf{K}^{-1}\left(\frac{d\mathbf{f}}{d\boldsymbol{\rho}} - \frac{d\mathbf{K}}{d\boldsymbol{\rho}}\boldsymbol{\alpha}\right) + \left.\frac{\partial g}{\partial \boldsymbol{\rho}}\right|_{\substack{\boldsymbol{\sigma}(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \tag{7.25}$$

The adjoint variable $\boldsymbol{\lambda}_j$ that gives the name to this procedure, can be defined at this point as:

$$\boldsymbol{\lambda_j}^T = \left.\frac{\partial g}{\partial \boldsymbol{\sigma}}\right|_{\substack{\boldsymbol{\sigma}(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\boldsymbol{\sigma}}{d\boldsymbol{\alpha}}\right|_{\boldsymbol{\alpha}(\boldsymbol{\rho})} \mathbf{K}^{-1} \tag{7.26}$$

In this way, the value of the adjoint variable can be obtained by solving the system of linear equations:

$$\mathbf{K}^T\boldsymbol{\lambda}_j = \left.\left(\frac{d\boldsymbol{\sigma}}{d\boldsymbol{\alpha}}\right)^T\right|_{\boldsymbol{\alpha}(\boldsymbol{\rho})} \left.\left(\frac{\partial g}{\partial \boldsymbol{\sigma}}\right)^T\right|_{\substack{\boldsymbol{\sigma}(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \tag{7.27}$$

Thus once the system of equations is solved, the value of the adjoint variable $\boldsymbol{\lambda}_j$ will be obtained, and the expression of the first order derivative of the damage constraint consists on:

$$\frac{dg}{d\boldsymbol{\rho}} = \boldsymbol{\lambda}_j^T\left(\frac{d\mathbf{f}}{d\boldsymbol{\rho}} - \frac{d\mathbf{K}}{d\boldsymbol{\rho}}\boldsymbol{\alpha}\right) + \left.\frac{\partial g}{\partial \boldsymbol{\rho}}\right|_{\substack{\boldsymbol{\sigma}(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \tag{7.28}$$

Finally, since the Galerkin approach is used, structural stiffness matrix will be always symmetrical and the matrix of (7.27) will be also the stiffness matrix. As a result, the sensitivity analysis only requires the calculation of the independent term and the use of the same factorization that had been used for the structural analysis in order to obtain the adjoint variable.

Apart from that, the first order directional derivatives can be calculated through the multiplication of the first order derivative vector and the improvement design direction.

7.4. Sensitivity analysis of the objective function

The sensitivity analysis of the objective function will be obtained through the general formula proposed in Chapter 5, (5.11), considering the penalization of the intermediate values of density.

Therefore, it is possible analytically to directly obtain the first order derivative of the objective function, since it does not depend on the state variables.

On the other hand, it will be necessary to introduce in the formulation that will be developed next, the value of the elemental volume.

Hence, the first order derivatives of the objective function can be obtained as:

$$\text{FEM} \quad \frac{\partial F}{\partial \rho_i} = \sum_{e=1}^{e=n} \frac{\partial F}{\partial \rho_e} \frac{d\rho_e}{d\rho_i} \qquad \text{IGA} \quad \frac{\partial F}{\partial \rho_i} = \sum_{e=1}^{e=n} \frac{\partial F}{\partial \rho_e} \frac{d\rho_e}{d\rho_i} \qquad i = 1, ..., n \tag{7.29}$$

where ρ_i are the design variables and the derivative of the objective function with respect to the elemental density can be calculated as:

$$\text{FEM} \quad \frac{dF}{d\rho_e} = \frac{1}{p} \rho_e^{\left(\frac{1}{p}-1\right)} V_e \qquad \text{IGA} \quad \frac{dF}{d\rho_e} = \frac{1}{p} \rho_e^{\left(\frac{1}{p}-1\right)} V_{ks} \qquad e = 1, ..., N_e \tag{7.30}$$

where p is the penalization factor of intermediate densities whose effect has been explained in Chapter 5.

Moreover, the other term of the first order derivative of the objective function has been calculated previously in (7.9), if the Finite Element Method is used or (7.10) in case of using the Isogeometric Analysis.

Finally, the first order directional derivative of the objective function can be calculated through the multiplication of its first order derivative and the improvement design direction in both cases.

7.5. Sensitivity analysis of the damage constraint

Once the procedure to obtain the sensitivity analysis of functions that depends on the state equation has been introduced, the calculation of the sensitivity analysis of the damage constraint is straightforward.

Although the development of the formulation of damage constraint has been made in Chapter 5, it will be also necessary to involve the structural analysis developed in Chapter 4.

Moreover, the displacement values $\left(\mathbf{u}^h(\boldsymbol{\rho})\right)$ has been the magnitude used to establish all the formulations and developments of the structural analysis. On the other hand, the stress values $\left(\boldsymbol{\sigma}^h(\mathbf{u})\big|_{\mathbf{u}=\mathbf{u}^h(\boldsymbol{\rho})}\right)$ can be calculated with the displacements value for both methods the Finite Element Method and the Isogeometric Analysis.

However, in order to simplify the notation, the stress tensor and the displacement distribution will be represented by $\boldsymbol{\sigma}$ and $\mathbf{u}$ respectively and the nodal displacement vector computed in the structural analysis will be represented by $\boldsymbol{\alpha}$.

7.5.1. First order derivatives

The first order derivative of the damage constraint can be calculated through the use of the adjoint variable approach, (7.26) and (7.28). Moreover, the equations that will have to be taken into consideration in the calculation of the first order derivative of the damage constraint will be:

$$g(\boldsymbol{\rho}) = \sum_{s=1}^{N_{str}} \left(\frac{\tilde{\rho}_s(\boldsymbol{\rho}, \boldsymbol{\sigma}_s(\boldsymbol{\rho}))}{\rho_s(\boldsymbol{\rho})} - 1 \right) \leq 0 \tag{7.31}$$

where $\tilde{\rho_s}$ and ρ_s are the value of the density in the point where the stress is calculated in the damaged model and the original one, respectively. On the other hand, the value of the density in the damaged model can be calculated as:

$$\tilde{\rho_s} = \rho_{min} + \beta_s(\rho_s - \rho_{min}) \qquad \text{where} \qquad \beta_s(\sigma; \sigma_{lim}) \geq 1 \tag{7.32}$$

is the value of the penalization factor, that can be determined as:

$$\beta_s(x_s) = \begin{cases} 1, & \text{if} \quad x_s \leq 0.9 \\ e^{(7.672705\rho_s^q(x_s-0.9))^{20}} & \text{if} \quad 0.9 < x_s \leq 1 \\ e^{50(\rho_s^q(x_s-0.99))^2} & \text{if} \quad x_s > 1 \end{cases} \tag{7.33}$$

where x_s is the formulation of the stress criterion considered in the solution of the topology optimization problem, that has been formulated as:

$$x_s = \frac{\hat{\sigma}_{VM,s}(\boldsymbol{\sigma}_s^h(\boldsymbol{\rho}))}{\hat{\sigma}_{max}\varphi_s} \tag{7.34}$$

where φ_s is the stress relaxation coefficient and $\hat{\sigma}_{VM,s}$ is the Von Mises stress, since the Von Mises stress criterion has been the criterion chosen to impose the stress constraints. The formulation of the stress relaxation coefficient is:

$$\varphi_s = 1 - \varepsilon + \frac{\varepsilon}{\rho_s} \tag{7.35}$$

The Von Mises stress criterion can be written in two-dimensional problems as:

$$\begin{aligned}
&\hat{\sigma}_{VM,s}(\boldsymbol{\sigma}_s) = \sqrt{\sigma_{x,s}^2 + \sigma_{y,s}^2 - \sigma_{x,s}\sigma_{y,s} + 3\tau_{xy,s}^2} \qquad \text{Plane Stress} \\
&\hat{\sigma}_{VM,s}(\boldsymbol{\sigma}_s) = \sqrt{\sigma_{x,s}^2 + \sigma_{y,s}^2 + \sigma_{z,s}^2 - \sigma_{x,s}\sigma_{y,s} - \sigma_{x,s}\sigma_{z,s} - \sigma_{y,s}\sigma_{z,s} + 3\tau_{xy,s}^2} \\
&\text{Plane Strain}
\end{aligned} \tag{7.36}$$

and in three-dimensional problem it can be written as:

$$\begin{aligned}
&\hat{\sigma}_{VM,s}(\boldsymbol{\sigma}_s) = \\
&\sqrt{\frac{1}{2}\left[(\sigma_{x,s} - \sigma_{y,s})^2 + (\sigma_{y,s} - \sigma_{z,s})^2 + (\sigma_{z,s} - \sigma_{x,s})^2 + 6(\tau_{xy,s}^2 + \tau_{yz,s}^2 + \tau_{zx,s}^2)\right]}
\end{aligned} \tag{7.37}$$

and finally, the stress vector in each point where a stress constraint is imposed, will be, for each situation:

$$\text{2D} \begin{array}{c}\text{Plane}\\\text{Stress}\end{array} \ \boldsymbol{\sigma}_s = \begin{Bmatrix} \sigma_{x,s} \\ \sigma_{y,s} \\ \tau_{xy,s} \end{Bmatrix} \quad \text{2D} \begin{array}{c}\text{Plane}\\\text{Strain}\end{array} \ \boldsymbol{\sigma}_s = \begin{Bmatrix} \sigma_{x,s} \\ \sigma_{y,s} \\ \tau_{xy,s} \\ \sigma_{z,s} \end{Bmatrix} \quad \text{3D} \ \boldsymbol{\sigma}_s = \begin{Bmatrix} \sigma_{x,s} \\ \sigma_{y,s} \\ \sigma_{z,s} \\ \tau_{xy,s} \\ \tau_{yz,s} \\ \tau_{zx,s} \end{Bmatrix} \tag{7.38}$$

At this point, it will be necessary to take into consideration the relationship between the local density ρ_s and the design variables ρ_i established in (7.11) and (7.12).

Once all functions and considerations to take into account in the formulation have been collected, first order derivatives of the damage constraint are calculated as:

$$\frac{dg}{d\rho_i} = \left.\frac{\partial \hat{g}}{\partial \sigma_{VM,s}}\right|_{\substack{\sigma_{VM,s}(\boldsymbol{\sigma}_s)\\ \boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\sigma_{VM,s}}{d\boldsymbol{\sigma}_s}\right|_{\substack{\boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\boldsymbol{\sigma}_s}{d\boldsymbol{\alpha}}\right|_{\boldsymbol{\alpha}(\boldsymbol{\rho})} \frac{d\boldsymbol{\alpha}}{d\rho_i} + \left.\frac{\partial \hat{g}}{\partial \rho_i}\right|_{\substack{\sigma_{VM,s}(\boldsymbol{\sigma}_s)\\ \boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \tag{7.39}$$

where the first order derivative of the damage constraint can be expressed as a vector:

$$\frac{dg}{d\boldsymbol{\rho}} = \left\{ \frac{dg}{d\rho_i} \right\}_{i=1,\dots,n} \tag{7.40}$$

Furthermore, the equivalence $\frac{d\boldsymbol{\alpha}}{d\boldsymbol{\rho}}$ obtained in (7.24) can be introduced in (7.39) as:

$$\begin{aligned}
\frac{dg}{d\rho_i} = &\left.\frac{\partial \hat{g}}{\partial \sigma_{VM,s}}\right|_{\substack{\sigma_{VM,s}(\boldsymbol{\sigma}_s)\\ \boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\sigma_{VM,s}}{d\boldsymbol{\sigma}_s}\right|_{\substack{\boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\boldsymbol{\sigma}_s}{d\boldsymbol{\alpha}}\right|_{\boldsymbol{\alpha}(\boldsymbol{\rho})} \mathbf{K}^{-1} \left(\frac{d\mathbf{f}}{d\rho_i} - \frac{d\mathbf{K}}{d\rho_i}\boldsymbol{\alpha} \right) \\
&+ \left.\frac{\partial \hat{g}}{\partial \rho_i}\right|_{\substack{\sigma_{VM,s}(\boldsymbol{\sigma}_s)\\ \boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}}
\end{aligned} \tag{7.41}$$

Therefore, the adjoint variable can be defined according to the idea proposed in (7.26) as:

$$\boldsymbol{\lambda}^T = \left.\frac{\partial \hat{g}}{\partial \sigma_{VM,s}}\right|_{\substack{\sigma_{VM,s}(\boldsymbol{\sigma}_s)\\ \boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\sigma_{VM,s}}{d\boldsymbol{\sigma}_s}\right|_{\substack{\boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \left.\frac{d\boldsymbol{\sigma}_s}{d\boldsymbol{\alpha}}\right|_{\boldsymbol{\alpha}(\boldsymbol{\rho})} \mathbf{K}^{-1} \tag{7.42}$$

Thus, first order derivative can be calculated as:

$$\frac{dg}{d\rho_i} = \boldsymbol{\lambda}^T \left(\frac{d\mathbf{f}}{d\rho_i} - \frac{d\mathbf{K}}{d\rho_i}\boldsymbol{\alpha}\right) + \left.\frac{\partial \hat{g}}{\partial \rho_i}\right|_{\substack{\sigma_{VM,s}(\boldsymbol{\sigma}_s)\\ \boldsymbol{\sigma}_s(\boldsymbol{\alpha})\\ \boldsymbol{\alpha}(\boldsymbol{\rho})}} \tag{7.43}$$

Finally, once the general formulation of the first order derivative of the damage constraint has been developed, it is time to establish the value of the unknown terms.

First, the term $\frac{\partial g}{\partial \sigma_{VM,i}}$ will be developed. In this case, it will be necessary to consider equations (7.31), (5.21), (7.33) and (7.34), on the other hand, this term will have to be calculated through the chain rule as:

$$\frac{\partial g}{\partial \sigma_{VM,s}} = \frac{\partial g}{\partial \tilde{\rho_s}} \frac{\partial \tilde{\rho_s}}{\partial \beta_s} \frac{\partial \beta_s}{\partial x_s} \frac{\partial x_s}{\partial \sigma_{V\tilde{M},s}} \tag{7.44}$$

where the partial derivative of the damage constraint with respect to the relative density of the damaged model $\tilde{\rho_s}$ is:

$$\frac{\partial g}{\partial \tilde{\rho_s}} = \frac{1}{\rho_s(\boldsymbol{\rho})} \tag{7.45}$$

and the partial derivative of the relative density of the damaged model $\tilde{\rho_s}$ with respect to the damage coefficient β_s is:

$$\frac{\partial \tilde{\rho_s}}{\partial \beta_s} = (\boldsymbol{\rho}_s(\rho) - \rho_{min}) \tag{7.46}$$

and the partial derivative of the damage coefficient β_s with respect to the stress criterion considered in the formulation x_s is:

$$\frac{\partial \beta_s}{\partial x_s} = \begin{cases} 0, & \text{if} \quad x_s \leq 0.9 \\ 20e^{(7.672705\rho_s^q(x_s-0.9))^{20}}\rho_s^{20q}7.672705^{20}(x_s-0.9)^{19} & \text{if} \quad 0.9 < x_s \leq 1 \\ 2e^{50(\rho_s^q(x_s-0.99))^2}50\rho_s^{2q}(x_s-0.99) & \text{if} \quad x_s > 1 \end{cases} \tag{7.47}$$

and, finally, the partial derivative of the stress criterion x_s with respect to the Von Mises stress $\sigma_{VM,s}$ is:

$$\frac{\partial x_s}{\partial \sigma_{V\tilde{M},s}} = \frac{1}{\hat{\sigma}_{max}\varphi_s} \tag{7.48}$$

Then, the term $\frac{d\sigma_{VM,i}}{d\boldsymbol{\sigma}_i}$ will be analyzed, both for three-dimensional problems and for two-dimensional problems with plane stress and plane strain. Therefore, the sensitivity analysis of the Von Mises stress criterion (7.34) with respect to the stress tensor can be defined as:

$$\frac{d\sigma_{VM,s}}{d\boldsymbol{\sigma}_s} = \left(\frac{\partial\sigma_{VM,s}}{\partial\sigma_{x,s}} \quad \frac{\partial\sigma_{VM,s}}{\partial\sigma_{y,s}} \quad \frac{\partial\sigma_{VM,s}}{\partial\sigma_{z,s}} \quad \frac{\partial\sigma_{VM,s}}{\partial\tau_{xy,s}} \quad \frac{\partial\sigma_{VM,s}}{\partial\tau_{yz,s}} \quad \frac{\partial\sigma_{VM,s}}{\partial\tau_{zx,s}} \right) \tag{7.49}$$

In the three-dimensional problems the value of the first order derivatives are:

$$\begin{aligned}
\frac{\partial\sigma_{VM,s}}{\partial\sigma_{x,s}} &= \frac{2\sigma_{x,s} - \sigma_{y,s} - \sigma_{z,s}}{2\sigma_{VM,s}}; & \frac{\partial\sigma_{VM,s}}{\partial\tau_{xy,s}} &= \frac{6\tau_{xy,s}}{2\sigma_{VM,s}} \\
\frac{\partial\sigma_{VM,s}}{\partial\sigma_{y,s}} &= \frac{2\sigma_{y,s} - \sigma_{x,s} - \sigma_{z,s}}{2\sigma_{VM,s}}; & \frac{\partial\sigma_{VM,s}}{\partial\tau_{yz,s}} &= \frac{6\tau_{yz,s}}{2\sigma_{VM,s}} \\
\frac{\partial\sigma_{VM,s}}{\partial\sigma_{z,s}} &= \frac{2\sigma_{z,s} - \sigma_{x,s} - \sigma_{y,s}}{2\sigma_{VM,s}}; & \frac{\partial\sigma_{VM,s}}{\partial\tau_{zx,s}} &= \frac{6\tau_{zx,s}}{2\sigma_{VM,s}}
\end{aligned} \tag{7.50}$$

In the two-dimensional problems with plane stress these derivatives are reduced to:

$$\begin{aligned}
\frac{\partial\sigma_{VM,s}}{\partial\sigma_{x,s}} &= \frac{2\sigma_{x,s} - \sigma_{y,s}}{2\sigma_{VM,s}} & \frac{\partial\sigma_{VM,s}}{\partial\tau_{xy,s}} &= \frac{6\tau_{xy,s}}{2\sigma_{VM,s}} \\
\frac{\partial\sigma_{VM,s}}{\partial\sigma_{y,s}} &= \frac{2\sigma_{y,s} - \sigma_{x,s}}{2\sigma_{VM,s}} & \frac{\partial\sigma_{VM,s}}{\partial\sigma_{z,s}} &= \frac{\partial\sigma_{VM,s}}{\partial\tau_{yz,s}} = \frac{\partial\sigma_{VM,s}}{\partial\tau_{zx,s}} = 0
\end{aligned} \tag{7.51}$$

And finally, in the two-dimensional problems with plane strain the derivatives are:

$$\begin{aligned}
\frac{\partial\sigma_{VM,s}}{\partial\sigma_{x,s}} &= \frac{2\sigma_{x,s} - \sigma_{y,s} - \sigma_{z,s}}{2\sigma_{VM,s}} & \frac{\partial\sigma_{VM,s}}{\partial\tau_{xy,s}} &= \frac{6\tau_{xy,s}}{2\sigma_{VM,s}} \\
\frac{\partial\sigma_{VM,s}}{\partial\sigma_{y,s}} &= \frac{2\sigma_{y,s} - \sigma_{x,s} - \sigma_{z,s}}{2\sigma_{VM,s}} & \frac{\partial\sigma_{VM,s}}{\partial\tau_{yz,s}} &= 0 \\
\frac{\partial\sigma_{VM,s}}{\partial\sigma_{z,s}} &= \frac{2\sigma_{z,s} - \sigma_{x,s} - \sigma_{y,s}}{2\sigma_{VM,s}} & \frac{\partial\sigma_{VM,s}}{\partial\tau_{zx,s}} &= 0
\end{aligned} \tag{7.52}$$

Then, the term $\frac{d\boldsymbol{\sigma}_i}{d\boldsymbol{\alpha}}$ will be calculated. In order to compute the sensitivity analysis of the stress tensor $\boldsymbol{\sigma}_i(\boldsymbol{\alpha})$ with respect to the nodal displacements $\boldsymbol{\alpha}$, it will be necessary to use these equations (4.2) and (4.21). In general, the combination of the equations previously mentioned provides the stress tensor formula in wherever point of the domain. This stress tensor can be calculated as:

$$\boldsymbol{\sigma}(\mathbf{r}^0) = \mathbf{DLu}(\mathbf{r}^0) \tag{7.53}$$

where $\mathbf{u}(\mathbf{r}^0)$ are the displacements of the point $\mathbf{r}^0$ considered. According to the discretization proposed in the structural analysis, it will be also possible to calculate the stress tensor $\boldsymbol{\sigma}(\mathbf{r}^0)$ by means of the nodal displacements $\boldsymbol{\alpha}$ as:

$$\boldsymbol{\sigma}(\mathbf{r}^0) = \sum_{i=1}^{N} \mathbf{DL\Phi}_i(\mathbf{r}^0)\boldsymbol{\alpha}_i \tag{7.54}$$

where N is the number of nodes that are necessary to solve the structural analysis and $\mathbf{\Phi}_i(\mathbf{r}^0)$ is the shape function matrix in the point $\mathbf{r}^0$,

$$\boldsymbol{\alpha} = \{\boldsymbol{\alpha}_i\}_{i=1,\dots,N} \qquad \text{and} \qquad \boldsymbol{\alpha}_i = \begin{Bmatrix} \alpha_{i1} \\ \vdots \\ \alpha_{id} \end{Bmatrix} \tag{7.55}$$

being d the number of dimensions of the problem. In this thesis, two-dimensional ($d = 2$) and three-dimensional problems ($d = 3$) will be solved.

Since the material constitutive matrix $\mathbf{D}$, the differential operator matrix $\mathbf{L}$ and the shape functions matrix $\mathbf{\Phi}_i$ in the point $\mathbf{r}^0$ do not depend on the design variables. The term $\frac{d\boldsymbol{\sigma}_s}{d\boldsymbol{\alpha}}$ can be calculated as:

$$\begin{aligned}
&\text{2D} \quad \frac{d\boldsymbol{\sigma}}{d\alpha_{i1}} = \mathbf{DL\Phi}_i(\mathbf{r}^0)\begin{pmatrix}1\\0\end{pmatrix} \quad \frac{d\boldsymbol{\sigma}}{d\alpha_{i2}} = \mathbf{DL\Phi}_i(\mathbf{r}^0)\begin{pmatrix}0\\1\end{pmatrix} \quad i = 1, ..., N \\
&\text{3D} \quad \frac{d\boldsymbol{\sigma}}{d\alpha_{i1}} = \mathbf{DL\Phi}_i(\mathbf{r}^0)\begin{pmatrix}1\\0\\0\end{pmatrix} \quad \frac{d\boldsymbol{\sigma}}{d\alpha_{i2}} = \mathbf{DL\Phi}_i(\mathbf{r}^0)\begin{pmatrix}0\\1\\0\end{pmatrix} \quad \frac{d\boldsymbol{\sigma}}{d\alpha_{i3}} = \mathbf{DL\Phi}_i(\mathbf{r}^0)\begin{pmatrix}0\\0\\1\end{pmatrix}
\end{aligned} \tag{7.56}$$

As it was proposed in (4.4), the matrix $\mathbf{\Phi}_i(r^0)$ can be obtained for two-dimensional and three-dimensional problems as:

$$\begin{aligned}
&\text{2D} \qquad \mathbf{\Phi}_i(\mathbf{r}^0) = \phi_i(\mathbf{r}^0)\mathbf{I}_2 \\
&\text{3D} \qquad \mathbf{\Phi}_i(\mathbf{r}^0) = \phi_i(\mathbf{r}^0)\mathbf{I}_3
\end{aligned} \tag{7.57}$$

where ϕ_i is the shape function of the node i calculated in the point $\mathbf{r}^0$.

Finally, the term $\frac{\partial g}{\partial \rho_i}$ will be calculated through the use of the chain rule. Equations (5.21), (7.31), (7.33) and (7.34), will be considered in this calculation. Furthermore, this derivative will be:

$$\frac{\partial g}{\partial \rho_i} = \frac{\partial \hat{g}}{\partial \rho_s}\frac{\partial \rho_s}{\partial \rho_i} + \frac{\partial \hat{g}}{\partial \tilde{\rho}_s}\left(\frac{\partial \tilde{\rho}_s}{\partial \rho_s}\frac{\partial \rho_s}{\partial \rho_i} + \frac{\partial \tilde{\rho}_s}{\partial \beta_s}\left(\frac{\partial \beta_s}{\partial \rho_s}\frac{\partial \rho_s}{\partial \rho_i} + \frac{\partial \beta_s}{\partial x_s}\frac{\partial x_s}{\partial \varphi_s}\frac{d\varphi_s}{d\rho_s}\frac{\partial \rho_s}{\partial \rho_i}\right)\right) \tag{7.58}$$

Although some of these terms have been calculated previously, they are all summarized now as:

$$\frac{\partial g}{\partial \rho_s} = -\frac{\tilde{\rho}_s(\boldsymbol{\rho}, \boldsymbol{\sigma}_s(\boldsymbol{\rho}))}{(\rho_s(\boldsymbol{\rho}))^2} \tag{7.59}$$

$$\frac{\partial g}{\partial \tilde{\rho_s}} = \frac{1}{\rho_s(\boldsymbol{\rho})} \tag{7.60}$$

$$\frac{\partial \tilde{\rho_s}}{\partial \rho_s} = \beta_s \tag{7.61}$$

$$\frac{\partial \tilde{\rho_s}}{\partial \beta_s} = (\rho_s - \rho_{min}) \tag{7.62}$$

$$\frac{\partial \beta_s}{\partial \rho_s} = \begin{cases} 0, & \text{if} \quad x_s \leq 0.9 \\ 20qe^{(7.672705\rho_s^q(x_s-0.9))^{20}}\rho_s^{20q-1}7.672705^{20}(x_s-0.9)^{20} & \text{if} \quad 0.9 < x_s \leq 1 \\ 2qe^{50(\rho_s^q(x_s-0.99))^2}50\rho_s^{2q-1}(x_s-0.99)^2 & \text{if} \quad x_s > 1 \end{cases} \tag{7.63}$$

$$\frac{\partial \beta_s}{\partial x_s} = \begin{cases} 0, & \text{if} \quad x_s \leq 0.9 \\ 20e^{(7.672705\rho_s^q(x_s-0.9))^{20}}\rho_s^{20q}7.672705^{20}(x_s-0.9)^{19} & \text{if} \quad 0.9 < x_s \leq 1 \\ 2e^{50(\rho_s^q(x_s-0.99))^2}50\rho_s^{2q}(x_s-0.99) & \text{if} \quad x_s > 1 \end{cases} \tag{7.64}$$

$$\frac{\partial x_s}{\partial \varphi_s} = -\frac{\hat{\sigma}_{VM,s}(\boldsymbol{\sigma}_s^h(\boldsymbol{\rho}))}{\hat{\sigma}_{max}\varphi_s^2} \tag{7.65}$$

$$\frac{d\varphi_s}{d\rho_s} = -\frac{\varepsilon}{\rho_s^2} \tag{7.66}$$

7.6. Overview

The sensitivity analysis is a fundamental part in the solution of topology optimization problems of structures by means of mathematical programming, because this information is the basis of optimization algorithms operation.

Furthermore, the computation of the sensitivity analysis generally means a big percentage of the computational effort required to deal with the topology optimization problem, both in terms of computational time and data storage.

Therefore, an analytical differentiation approach has been developed in this thesis in order to calculate the first order derivatives. Moreover, this procedure has demonstrated to be efficient both in CPU time and data storage.

The first order derivatives of the damage constraint are calculated with an analytical differentiation algorithm by means of the adjoint variable approach, that offers many computational advantages. However, the sensitivity analysis of the objective function can be made with direct analytical differentiation techniques.

In this chapter, a special emphasis over the computational requirements had been made. The main reason is that they will be very important, especially in topology optimization problems of structures with high spatial definition. In this kind of problems, a big amount of design variables is required to obtain high quality results. As a result, a big amount of derivatives will have to be calculated also.

Finally, a flowchart of the topology optimization of structures algorithm will be developed in the next chapter. Additionally, some implementation aspects that has been introduced in the algorithm in order to improve its efficiency will be analyzed.

Chapter 8

Numerical implementation

"Bad programmers worry about the code. Good programmers worry about data structures and their relationships"
Linus Torvalds, (1969-XXXX).

8.1. Introduction

Although the whole theoretical development of the Topology Optimization Problem of minimum weight and Damage Constraint has been made in the previous chapters, its numerical analysis has not been made yet. For this reason, the analysis of the numerical implementation and its computational requirements will be developed in this chapter.

The way calculations will be made in the numerical implementation of the topology optimization algorithm will be different depending on the method used to solve the problem. Two alternative approaches have been implemented, the Finite Element Method and the Isogeometric Analysis, since they have different pros and cons.

The structural analysis models and the material layouts are different for both formulations. Both aspects will be analyzed in this chapter. On the other hand, the implementation of the method for two-dimensional or three-dimensional problems does not require any different analysis.

For all these reasons, at first the flowchart of the numerical implementation of the algorithm will be developed. Then, it has been necessary to introduce an adjustment of the maximum variation of the design variables in each iteration due to the high non-linearity of the damage constraint. Furthermore, the value of the penalization coefficient and the stress relaxation coefficient will have to be also modified during the solution of the topology optimization problem in order to obtain full-empty solutions and to ensure the appearance of areas with relative density equal to its lower limit respectively.

Therefore, this chapter will be structured as follows: first, the flowchart of the numerical implementation will be briefly analyzed. Then some parameters of the model are studied and adequately adjusted. Finally, some computational considerations such as the CPU time and computational requirements in the solution of the topology optimization problems will be analyzed.

8.2. Algorithm's flowchart

Once the topology optimization problem has been completely formulated, the flowchart of the topology optimization algorithm will be presented. The general scheme of the flowchart will be similar regardless the method used, however, some considerations will be taken into account. These little differences are related to the calculation of the shape functions and the points where stresses will be calculated. On the other hand, the differences between the use of the Finite Element Method and the Isogeometric Analysis will be mentioned in each step. Figure 8.1 shows a general flowchart of the algorithm. The general phases of this procedure are:

1. Preprocessing and Data Input: This is the first step of the topology optimization algorithm. In this phase the structural problem has to be completely defined (loads, geometry, number of elements, method used to solve it, ...). On the other hand, the initial value of the design variables and the general parameters that control the algorithm (Damage Coefficients, Penalty Coefficient ...) have to be established at this point. Finally, the shape functions are defined for the Finite Element Method, due to the constant value in the Gauss Points for all the elements of the mesh.

2. Structural Analysis: Once all the parameters and the design variables have been defined, the structural analysis is made. For this purpose, the Finite Element Method or the Isogeometric Analysis can be used. The method used has been described in Chapter 4. Additionally, the shape functions will have to be calculated for the Isogeometric Analysis. The results obtained in this step let calculate the field of structural displacements.

3. Objective Function: The objective function is calculated when the structural analysis is made. Its calculation has been formulated in Chapter 5 for both methods (Isogeometric Analysis and Finite Element Method). In this case, the value of the penalization coefficient of intermediate value of densities will have to be taken into account, since this value will be modified during the solution of the problem in order to obtain full-empty solutions.

4. Structural Stresses: The value of the stresses will be calculated when the structural analysis has been completed and the field of structural displacements is known. The Von Mises stress required to define the Damaged Model will be calculated by means of these stresses. At this point, there is a difference depending on the method chosen for solving the optimization problem. For the Finite Element Method, stresses have to be only calculated in the central point of each element, and for the Isogeometric Analysis some additional points will have to be incorporated. On the other hand,

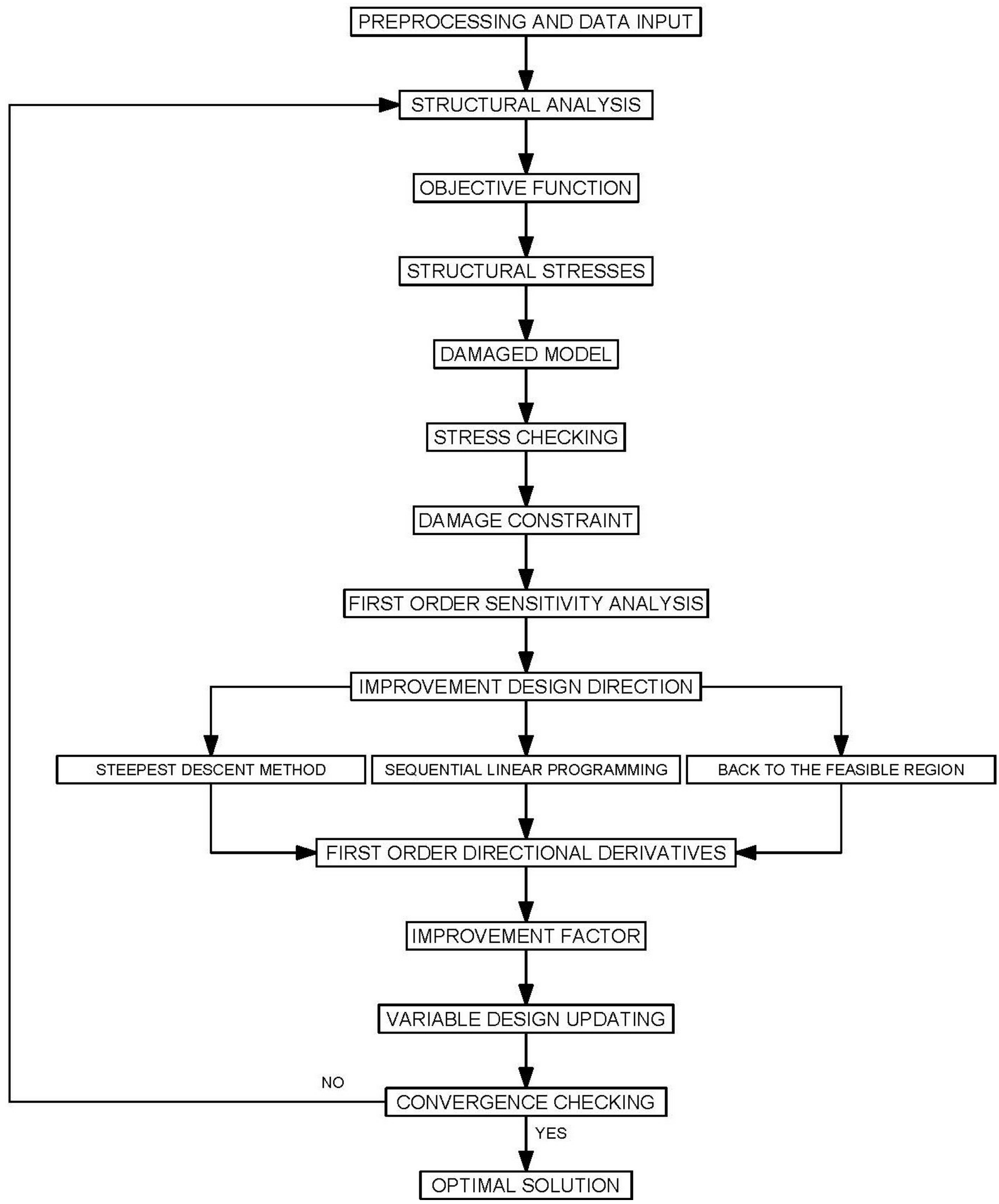


Figure 8.1. Solution methodology for Topology Optimization problems

for this purpose, the shape functions in case of Isogeometric Analysis will have to be calculated again.

5. Damaged Model: The damaged model can be generated when the Von Mises stresses are known by means of the penalization of the structural weight in the areas where

the stresses overcome their maximum allowable value. All the details about this model have been established in Chapter 5. The characteristic parameters of the model have been stated in the first step.

6. Stress checking: Although it is not strictly necessary the Von Mises stresses will be checked through the comparison with their maximum allowable value in order to have additional information about the reliability of the designs.

7. Damage Constraint: The value of the damage constraint can be calculated when the Damaged Model has been completely defined and its feasibility can be checked.

8. First Order Sensitivity Analysis: The first order sensitivity analysis need to be calculated for the objective function and the damage constraint. This sensitivity analysis has been developed in Chapter 7, where the equations of the derivatives have been established. On the other hand, the shape functions will have to be calculated again in case of using the Isogeometric Analysis.

9. Improvement design direction: The results obtained in the analysis of the damage constraint will determine the feasibility of the design. Then, the improvement design direction could be calculated. For this purpose, all the optimization methods developed in Chapter 6 can be used.

10. First order directional derivatives: The first order directional derivative of the objective function and the damage constraint will be calculated when the improvement design direction is known. These analysis requires a scalar product between the vector of derivatives and the improvement direction vector previously calculated.

11. Improvement Factor: The next step after computing the first order directional derivatives of the objective function and the damage constraint is to analyze the value of the improvement factor. For this purpose, the method described in Chapter 6 will be used.

12. Update of the design variables: Once the improvement factor has been calculated the value of all the design variables will be updated. In addition, an extra check of the side constraints is developed too.

13. Convergence Checking: Once all the design variables have been updated, the convergence checking is made by comparing two consecutive designs. In case that the difference between two consecutive designs can be considered negligible convergence is achieved. If the value of the damage constraint is extremely high the algorithm is stopped.

14. Writing Results: The results obtained during each iteration will be written at the end of the process in different files (design variables, stresses, CPU time, ...).

8.2.1. Evolutionary parameters

According to the considerations established in the previous chapters, the design methodology used to solve the topology optimization problems has to deal with three drawbacks: the high non-linearity of the objective function and the damage constraint, the search of full-empty solutions and the desire of achieving a high spatial definition.

The use of the damage model as a global approach makes possible the attainment of high spatial definition solutions. For this purpose, only one constraint, the damage constraint, has to be established regardless the number of design variables defined. However, the way this constraint is defined in this thesis will be kept unchanged during the solution procedure.

On the other hand, the introduction of a penalty coefficient in the objective function has been formulated in Chapter 5. This technique will be used to obtain full-empty solutions, because the intermediate values of density will be penalized. However, it will not be advisable to penalize intermediate densities in the first iterations of the procedure since a reduction of the non-linearity of the problem is intended. For this reason, the value of this coefficient and its range of application will vary according to table 8.1.

Iteration range	Penalty coefficient
(0)-(0.2n-1)	1
(0.2n)-(0.4n-1)	2
(0.4n)-(0.6n-1)	3
(0.6n)-(0.8n-1)	4
(0.8n)-(n)	5

Table 8.1. Value of the penalty coefficient

where n is the number of iterations required in the solution of the topology optimization problem.

The main reason to progressively increase the value of the penalty coefficient instead of using only two different values during the optimization procedure is to ensure the attainment of full-empty solutions. If only two different values of the penalty coefficient were considered during the optimization procedure, the main problem would be to choose the most appropriated values of this coefficient.

As it was mentioned above, it is advisable to avoid penalization of intermediate densities in the first iterations of the procedure. Apart from the reduction of the non-linearity of the problem, this circumstance is also due to the dependence of the solutions obtained with respect to the initial design, especially if the relative density does not take a constant value in all the domain. As a result of this, the first value chosen for the penalty coefficient should be 1.

On the other hand, the use of only one value different to 1 as a penalty coefficient could not be sufficient to attain solutions clearly defined. A vagueness phenomenon what means the existence of areas that are not clearly defined tends to appear in critical areas. These areas are in the proximity of the structural supports, in the surroundings of the points where loads are applied and in points where there is stress concentration. The use of different values of the penalty coefficient greater than 1 can deal with this phenomenon.

On the other side, the introduction of the stress relaxation coefficient in the stress constraint has been formulated in Chapter 5. This technique will be used to avoid singularity phenomena what will make possible the appearance of areas with relative density equal to its lower limit, since the maximum allowable stress will be slightly increased as the relative density is diminished. However, it will not be advisable to relax stresses in the first iterations of the process. For this reason, the value of this coefficient and its range of application will be defined in table 8.2.

Iteration range	Stress relaxation coefficient
(0)-(0.2n-1)	0
(0.2n)-(0.4n-1)	0.001
(0.4n)-(0.6n-1)	0.002
(0.6n)-(0.8n-1)	0.003
(0.8n)-(n)	0.005

Table 8.2. Value of the stress relaxation coefficient

where n is the number of iterations required in the solution of the topology optimization problem.

The main reason to increase progressively the value of the stress relaxation coefficient, instead of using only two different values, during the optimization process is to avoid sharp changes between a null stress relaxation and an excessive relaxation. This situation will happen when only two different values of the stress relaxation coefficient are considered during the optimization process. Moreover, the main problem would be also in this case, to choose the most appropriated values of this coefficient.

As it was mentioned above, it is not advisable to relax stresses in the first iteration of the process, since this circumstance tends to facilitate the appearance of areas with relative density equal to its lower limit and do not allow to readapt the designs. In other words, if the stresses are relaxed from the first iteration of the process, there can be parts of the structure that will not be perfectly defined during the optimization procedure because of an excessive stress relaxation at the beginning of the process. This removed parts can not appear again later if necessary.

On the other hand, the own definition of the damage constraint implies that the stresses can take whatever value if the relative density is equal to its lower limit, what means that it will be difficult to correct the effect of an excessive relaxation in the first iterations of the process. This phenomenon can be avoided if the stress relaxation coefficient is not introduced in the stress constraint until the structure has had time to develop its main topology properly, and then the stress relaxation coefficient can be introduced to allow the existence of areas with relative density equal to its lower limit (full-empty solutions).

Finally, the high non-linearity of the objective function and the damage constraint can produce the oscillation of the algorithm around the optimal solution. This circumstance can suppose that the algorithm never achieves the optimal solution and in some cases oscillates between two different solutions, one valid and other usually not valid. Due to this a reduction of the maximum modification of the design variables per iteration (moving limits) will be introduced at each iteration. However, the size of this moving limits can be changed at a certain number of iterations. In this case, this maximum modification is linearly reduced to 75% of the previous range each $0.1n$ iterations, being n the number of iterations required in the solution of the topology optimization problem.

8.3. Computational considerations

Finally, all the different phases of the optimization process from a computational point of view will be analyzed. For this purpose, it will be important to analyze the memory requirements and the CPU time, or in other words the number of floating point operations in each phase of the optimization process.

This analysis will allow to compare the computational requirements of each step and will make possible also to establish the critical point of the optimization algorithm from a computational point of view. The implementation of both the Finite Element Method and the Isogeometric Analysis allows to develop a comparative analysis of both methods.

Moreover, the differences between two-dimensional and three-dimensional problems will be established for the phases where they mean an important change of computational requirements. Then, a more specific analysis of the most relevant parameters or aspects for the structural analysis and the sensitivity analysis in terms of computational requirements will be made.

8.3.1. Structural analysis

One of the most important steps of the Topology Optimization Problem in terms of computational requirements is the structural analysis since it is computed at every iteration. In this thesis, the structural analysis can be solved by means of the Finite Element Method or the Isogeometric Analysis.

On the other hand, the value of the relative density will have influence in the calculation of the contribution of each element over the global stiffness matrix. However, this circumstance does not suppose an important drawback regarding to the computational requirements.

The system of linear equations that defines the structural problem is solved by means of the Cholesky Factorization, since the factorization of the matrix will have to be computed only once for solving other systems with the same matrix. This circumstance is important because this factorization will be used again in the sensitivity analysis.

Moreover, the factorized matrix is stored over the original stiffness matrix, and as a result, it will be necessary to store only the triangular matrix and the diagonal one obtained in the factorization. Finally, once the stiffness matrix is factorized, the system of linear equations can be solved.

The stiffness matrix will be stored in the computer memory by a Sky-line way in only one vector since the global stiffness matrix is a band matrix and this is the most efficient way to store this kind of matrixes. As a consequence, a remarkable reduction of CPU time and memory requirements is obtained with these techniques when the system of linear equations is solved.

Nevertheless, it will be necessary to take other considerations into account when the geometry of the mesh is defined. The way to enumerate the structural nodes is important, since a bad organization of them supposes an increase of the bandwidth.

	Finite Element Method (Quadratic serendipity elements)	Isogeometric Analysis (Quadratic elements)
2D	$3xy + 2(x{+}y) + 1$	$xy + 2(x{+}y) + 4$
3D	$4xyz + 3(xy{+}xz{+}yz) + 2(x{+}y{+}z) + 1$	$xyz + 2(xy{+}xz{+}yz) + 4(x{+}y{+}z) + 8$

Table 8.3. Number of points for structural analysis with regular meshes

The first aspect that will be commented in this thesis is the number of points used in the solution of the structural analysis. The number of points involved for regular meshes will be different depending on the method chosen. The general expressions to calculate the number of points for two-dimensional and three-dimensional problems can be observed in table 8.3 both for Finite Element Method and Isogeometric Analysis. In this expressions x, y and z are the number of elements in each dimension respectively.

For two-dimensional problems, the Isogeometric Analysis with quadratic elements only needs the 33% of the points required by the Finite Element Method with quadratic serendipity elements, this percentage is reduced until the 25% for the three-dimensional one.

This circumstance means an important save of computational requirements and CPU time if the Isogeometric Analysis is used for the structural analysis. This is due to the fact that the dimension of the general stiffness matrix is in this case the number of dimensions of the problem times the number of points. On the other hand, this also means that the system of linear equations will be smaller than the Finite Element Method one.

On the other hand, the way to enumerate the structural points only has influence in the computational requirements and the CPU time since the bandwidth depends on this enumeration. However, if the same enumeration strategy is used for both methods, there will not be important differences between them.

For all these reasons, the most advisable way to organize the points is to start with the dimension with the lowest number of elements. In case of three-dimensional problems, the rule should be used again in order to optimize the bandwidth of the matrix.

The order of the structural points is related to the bandwidth since the matrix obtained is a skyline matrix. Therefore, if the strategy to enumerate the nodes is different to the previously described, the number of matrix elements stored for solving the structural analysis will be significantly increased.

Even though, a deeper analysis of the variable bandwidth for each approach could be made. The general conclusion is that the variable bandwidth in the Isogeometric Analysis with quadratic elements will be always lower than in the Finite Element Method with quadratic serendipity elements.

The variable bandwidth can be calculated for both methods as the number of dimensions of the problem times the difference between the number of the node considered and the lowest number of node among the nodes that define the element or elements that are influenced by the node considered.

A quick way to demonstrate the difference in the bandwidth value between the Isogeometric Analysis and the Finite Element method, is to calculate the number of points required to define one file of elements in the dimension with the lowest number of elements, being n the number of elements in this direction. In the Isogeometric Analysis with quadratic elements, $3n + 6$ points are necessary to define it, and in case of the Finite Element Method with quadratic serendipity elements it is necessary to use $5n + 3$ points.

For all this, it is possible to conclude that the best way to solve the structural analysis, regarding to the CPU time and computational requirements, is by means of the Isogeometric Analysis. Or from another point of view, the same size of the general stiffness matrix allows to solve bigger problems with Isogeometric Analysis than with Finite Element Method.

8.3.2. Sensitivity analysis

The other important step in the solution of the topology optimization problem is the sensitivity analysis. In this thesis the first order derivatives will be calculated and they will have an important influence in the CPU time. For this purpose, the adjoint variable approach will be only used in case of the derivative of the damage constraint, since the derivative of the objective function has an expression easily computable. As a consequence, the computational requirements used for this step will not be considerably important, since only one constraint has been used in the formulation of the problem.

The only aspect that supposes a difference between the Finite Element Method and the Isogeometric Analysis regarding the computational requirements is the different number of design variables, that is slightly higher in the case of the Isogeometric

Analysis. However, this situation could be the opposite if other ways to define the material layout is established for both methods.

First, the sensitivity analysis of the objective function is one of the key points of the procedure, since it will have a big influence in the attainment of the improvement direction. The calculation of this derivative does not mean an important computational cost, since its analytical expression can be easily obtained and computed.

As it was mentioned above, the storage of the first order derivative vector of the objective function does not require a big cost, because their size will be equal to the number of design variables. Therefore, regardless the method used to solve the topology optimization problem, the CPU time and the computational requirements for this step will be negligible.

The next step is to analyze the sensitivity analysis of the damage constraint. This procedure will have a more important effect over the CPU time and the computational requirements than the sensitivity analysis of the objective function.

The computational requirements and the CPU time are related with the number of design variables used in the formulation of the problem, and they are considerably high especially in problems where a high spatial definition is intended.

As a consequence, the Adjoint Variable Approach will be used to calculate the first order derivatives of the damage constraint, since it will be necessary to solve only one system of linear equations and this is the most efficient method in terms of CPU time and computational requirements for this kind of problems.

Despite of all these improvements, the first order sensitivity analysis of the damage constraint is one of the most important steps in the topology optimization algorithm regarding the computational requirements.

At this point, it will be possible to make the comparative analysis between the use of the Finite Element Method and the Isogeometric Analysis. First, it will be important to consider the relationship between the elements of the mesh and the design variables.

In the Finite Element Method each element of the mesh usually depends on one design variable, however, in the Isogeometric Analysis each element (knot span) can be influenced by 9 or 27 design variables in case of two-dimensional or three-dimensional problems respectively.

The Finite Element Method will be most efficient in terms of computational requirements for the sensitivity analysis of the Objective Function and the Damage Constraint, but it is important to highlight that the material distribution is also C^0 in Finite Element Method, versus C^2 in Isogeometric Analysis.

8.3.3. Other parts of the method

Once the most important phases of the topology optimization problem in terms of computational requirements has been deeply analyzed. A brief analysis of the rest of the procedure will be developed.

The search of the best improvement direction is negligible in comparison to other algorithms since just one constraint is involved in the topology optimization problem, the Damage Constraint.

The computation of first order directional derivatives can be made directly with the first order derivatives by applying a scalar product with the improvement direction.

The computation of other parts of the algorithm do not depend on the method chosen. They will be related with different characteristics of the topology optimization problem.

First, the number of points that will define each element of the mesh. Then, the number of design variables employed for defining the material layout. And finally, the number of stresses to be considered in the definition of the Damage Constraint.

8.4. Overview

A specific methodology to solve topology optimization problems with minimum weight and the imposition of stress constraints by means of a damage constraint in two and three dimensions has been developed in this thesis. This methodology had been typically formulated for the Finite Element Method, and it had been an arduous task in terms of CPU time and computational requirements for high spatial definition problems.

Although the Simplex Algorithm was initially the critical point due to local formulation of the stress constraints, this problem was solved by means of the constraint aggregation techniques. However, the attainment of high spatial solution continues being an arduous task because of the computational requirements and the CPU time used for the structural analysis.

As a result of this, the Isogeometric Analysis has been implemented for solving the topology optimization problem with high spatial definition, since a little number of design variables in comparison to the conventional Finite Element Method to define the material layout provides solutions with the same spatial definition. Therefore, the Isogeometric Analysis has been introduced in the solution of the structural analysis apart from the definition of the material layout.

Even though, the main objective of the introduction of the Isogeometric Analysis was to define the material layout in a more efficient way in terms of design variables versus spatial definition, a reduction of the CPU time in the structural analysis step was also attained. However, an increase in the amount of CPU time needed for the sensitivity analysis was obtained due to the use of the Isogeometric Analysis to define the material layout.

In spite of this, the need of a lower number of design variables to obtain solutions with the same spatial definition with the Isogeometric Analysis compensates this increase of CPU time. For all these reasons, a deeper analysis of the most expensive steps (structural analysis and sensitivity analysis) has been made in this chapter.

In the structural analysis the emphasis will be on the number of points to be used and in the variable bandwidth of the matrix, since both aspects are related with the computational requirements.

In the sensitivity analysis, the key point is the spatial definition, the number of design variables used to define each element of the mesh. This supposes an important necessity of computational requirements, mainly CPU time.

Other parts and steps of the algorithm can be considered negligible in terms of computational requirements. Thus, the methodology proposed in this thesis for solving topology optimization of structures problems has been analyzed. The next step would be to solve some application examples with these models.

Chapter 9

Examples of application

"There is only one thing that makes a dream impossible to achieve: the fear of failure"
Paulo Coelho, (1947-XXXX).

9.1. Introduction

Once all the theoretical developments of the topology optimization problem of minimum weight with damage constraint have been made, several examples of application in the engineering field will be presented in this chapter.

In this thesis two-dimensional examples with plane stress and three-dimensional ones will be solved. Moreover, the linear elasticity theory with small displacements and small displacement gradients will be considered in both cases. Furthermore, all the examples developed will be solved with the Finite Element Method and the Isogeometric Analysis. On the other hand, the examples solved in this thesis will be structured in three different sections.

In the first section, some classical examples of the structural topology optimization will be used to validate the numerical formulation proposed in this thesis. In the second section, some practical examples in the two-dimensional space with plane stress will be solved in order to compare the solutions obtained with the solutions proposed previously by other authors.

At this point, it is important to remark that most of the solutions have been obtained by means of Finite Element Method. Therefore, the solutions obtained with the Isogeometric Analysis will have to be also compared with the solutions attained with Finite Element Method, since Isogeometric Analysis still has not been extensively used in the formulation of the topology optimization problem.

Finally, in the third section, the three-dimensional examples will be solved. In this case, it will not be necessary to make a testing of the numerical formulation since the structural analysis is the only part of the algorithm that presents differences with respect to the formulation used to solve the problem in the two-dimensional space. Furthermore, the solution of the topology optimization problem with stress constraints in the three-dimensional space has not been widely developed yet.

The two alternative models (Finite Element Method and Isogeometric Analysis) used to calculate the structural analysis has been introduced in Chapter 4. Moreover, the failure criterion chosen to introduce the effect of the structural stresses in the definition of the Damage Constraint has been the Von Mises criterion, since the material used in all the examples proposed will be steel.

Although other materials could be used to solve this kind of problem, this circumstance has not been a subject of study in this thesis, since the use of other materials would require the introduction of different failure criteria in the formulation of the Damage Constraint in the Topology Optimization Problem.

Conversely, the initial design used in this thesis to solve all the examples of application will have all the structural domain full of material. In other words, all the design variables of the topology optimization problem will start with their maximum allowable value. On the other hand, it will be important that the initial solution of the problem will be feasible. Otherwise, the optimization algorithm will initially look for a feasible solution of the problem, in case that they exist. And when this solution has been found, the algorithm will begin to optimize it.

This circumstance is due to the use of stress constraints in the formulation of the topology optimization problem of minimum weight, and as a result it will not be possible to guarantee that all the problems formulated will have a feasible solution. This situation tends to happen when the loads applied over the structural domain are excessive. This is the real situation in engineering practice involving design and structural analysis.

Additionally, the minimum value of the relative density used in all the examples proposed in this thesis will be $\rho_{min} = 0.001$. This is the typical value used in other works of the structural topology optimization field. The main reason to choose a non zero value is to avoid the singularity phenomenon of the elementary stiffness matrix of the structural analysis. However, it could be possible to choose smaller values of the minimum value of the relative density without introducing limitations to the numerical method.

Apart from that, it is important to remark that some of the parameters required by the formulations used in the solution of the topology optimization problem will have to be modified during this process, due to the high non-linearity of the objective function and the damage constraint used in the definition of the topology optimization problem.

Moreover, it will not be possible to ensure the attainment of the global optimum since the problem formulated is in general non convex. Nevertheless, the parameters used in the definition of the problem will have influence in the convexity of the problem and as a result the topology optimization algorithms will stop at any local optimum only if the value of all the parameters remain constant during all optimization process.

For this reason, it will be important to start with parameters that improves the convexity of the design area and reduce the non-linearity, with the objective of approximating to the global optimum solution. The search for essentially binary solutions and the stress relaxation to avoid singularity phenomena requires that some parameters will have to be modified during the optimization process.

The parameters modified during the solution of the topology optimization problem will be: the penalty coefficient of intermediate density values that will be initially $p = 1$ and it will be progressively increased, the stress relaxation coefficient that will be initially $\epsilon = 0$ since no singularity phenomena appears in the first iterations of the optimization procedure. However, its value will be increased during the optimization procedure. And finally the maximum modification of the design variables at each iteration that, in contrast to the previous parameters, will be reduced during the solution of the topology optimization problem in order to facilitate the convergence.

The penalty coefficient of intermediate density values will be modified in order to obtain full-void solutions, and the stress relaxation coefficient will be introduced to deal with the singularity phenomena when the density tends to zero, since in these cases there would not be nor material rather stress constraint.

On the other hand, the maximum modification of the design variables is modified to avoid the oscillation of the solution obtained around the optimal solution due to the high non-linearity of the objective function and the damage constraint and the first order Taylor expansions used. Even though the different values that these parameters will take during the solution of the topology optimization problem have been established in Chapter 8, they will be also commented in all the examples proposed.

The software applications used in this thesis to do all the calculation have been developed in Fortran. Although some parts of the topology optimization algorithm such as the sensitivity analysis can be calculated in parallel, its implementation has been developed in sequential mode.

The use of parallelization techniques would suppose that the CPU time required to solve the problem in case of the Isogeometric Analysis will be considerably reduced, especially for three-dimensional problems since sensitivity analysis is the critical part in terms of CPU time. However, the parallelization of the code would not introduce an important improvement regarding CPU time for the Finite Element Method, since the most constraining part is the structural analysis as it was commented in Chapter 8. Although, it could be possible to parallelize the construction of the global stiffness matrix, it would not be possible to parallelize the factorization of this matrix and the solution of the equations linear system.

Finally, the pictures shown in this thesis, has been obtained through the same software applications used to solve the topology optimization problem, that produces an image file in Paraview format. These image files have been created for the relative density and for the normalized stress, that is defined as the ratio between the value of the stress and the maximum allowable value of the stress at each point of the domain.

The graphic representation of the solution has not been submitted to any postprocessing or image filtering technique. All the values shown are directly obtained from the analysis model without post-processing.

9.2. Testing examples

First of all, it will be necessary to test the topology optimization algorithm developed in this thesis. For this purpose, the most common examples that have been solved in the topology optimization field will be solved in this section.

Therefore, the main objective of this section will be to validate the topology optimization algorithm, and particularly the use of the Isogeometric Analysis as a way to define the material layout and the use of the Damage Constraint to include stress constraints.

Three different examples will be solved in this section: a beam with large height and upper load, also known as arc solution, a cantilever beam and a L-shaped beam.

9.2.1. Beam with large height and upper load

The first example of this section is commonly known as arc structure. This example will have fixed supports in the lower corners what will mean vertical and horizontal displacements constrained on both sides. The main reason to include this example in this section is due to its extensive study in the topology optimization field, since its solution is known.

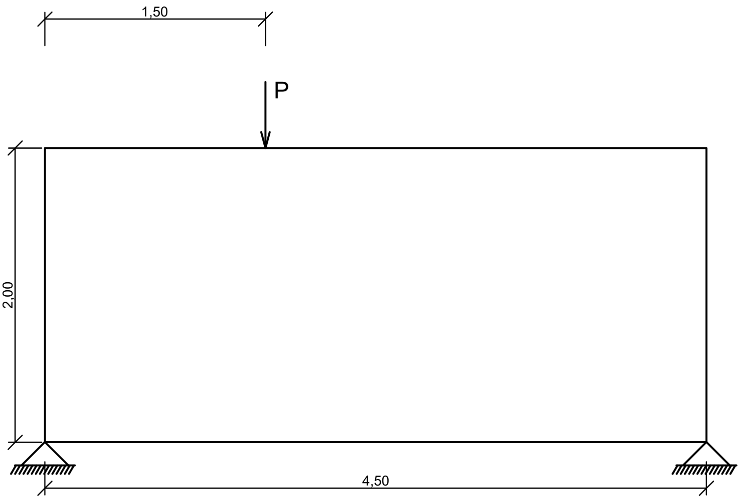


Figure 9.1. Beam with large height and upper load: Problem statement. (units - m)

First, figure 9.1 shows a graphical scheme with the dimensions of the domain of this problem and the position of the external loads applied. Moreover, it is important to establish that the structural weight will be considered as a structural load.

The structural domain will be discretized by means of a regular mesh of 225 x 100 = 22500 divisions of the domain in two-dimensions. These divisions will be quadratic serendipity elements with 8 nodes in the case of the Finite Element Method and quadratic knot spans with 9 nodes in the case of the Isogeometric Analysis. On the other hand, the structural thickness will be in this case 0.21 m.

Although the external load applied over the structure is a point load, this is not realistic in practice and it will be distributed over 12 adjacent elements in order to avoid the stress concentration phenomenon. The value of this point load will be equal to 40 10^3 kN. As it was previously commented, the material used for the design of this structure is steel with: material density $\gamma_{mat} = 7850\ kg/m^3$, Young's modulus $E = 2.1\ 10^5\ MPa$, Poisson's ratio $\nu = 0.3$ and yield stress $\tilde{\sigma}_{max} = 230\ MPa$.

Figures 9.2, 9.3, 9.4 and 9.5 show the evolution of the optimal design during the topology optimization process and figures 9.6 and 9.7 represent the final solution obtained with the formulations proposed in this thesis: the conventional Finite Element Method and the Isogeometric Analysis respectively.

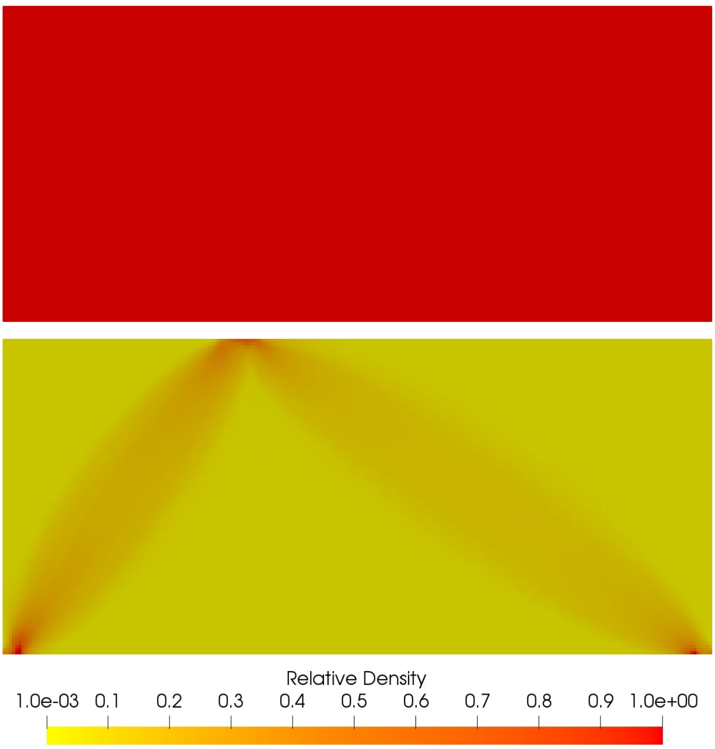


Figure 9.2. Beam with large height and upper load: Evolution of the relative density by using the Finite Element Method [iterations 0, 300]

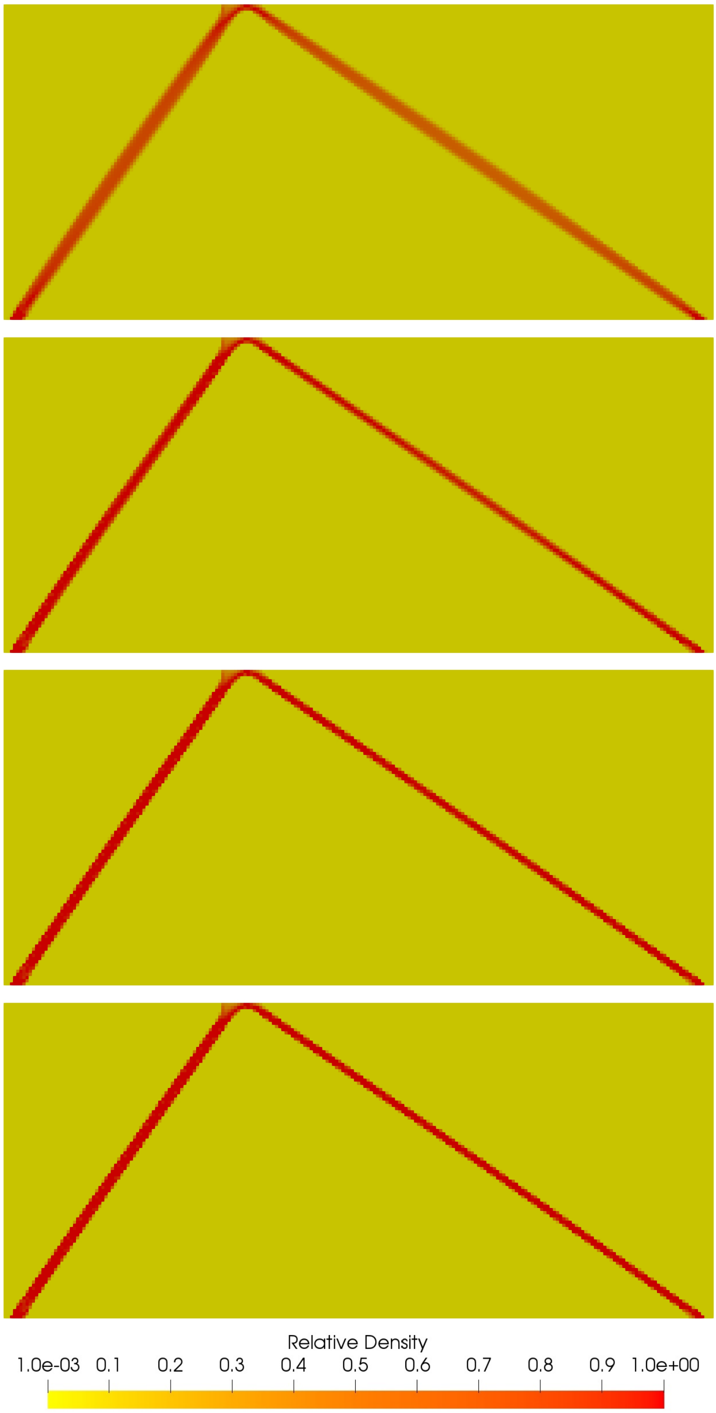


Figure 9.3. Beam with large height and upper load: Evolution of the relative density by using the Finite Element Method [iterations 600, 900, 1200, 1500]

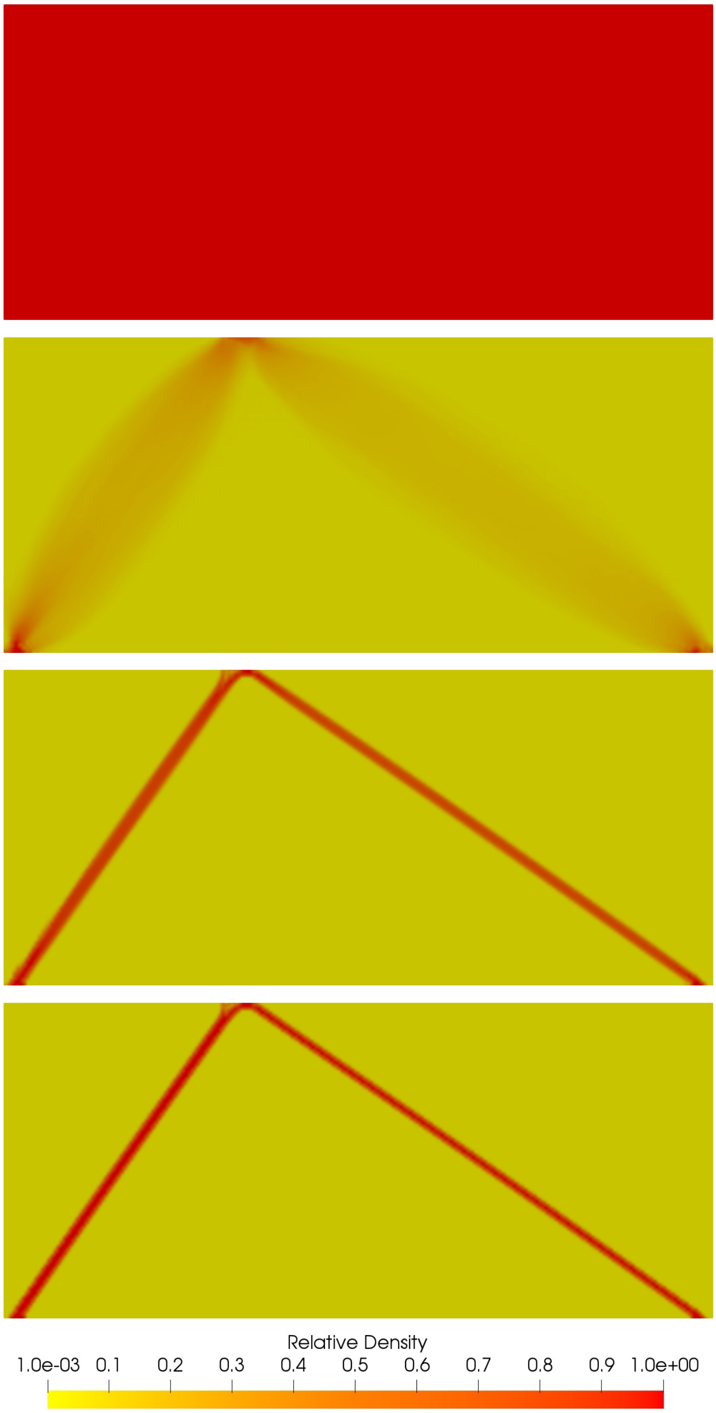


Figure 9.4. Beam with large height and upper load: Evolution of the relative density by using the Isogeometric Analysis [iterations 0, 300, 600, 900]

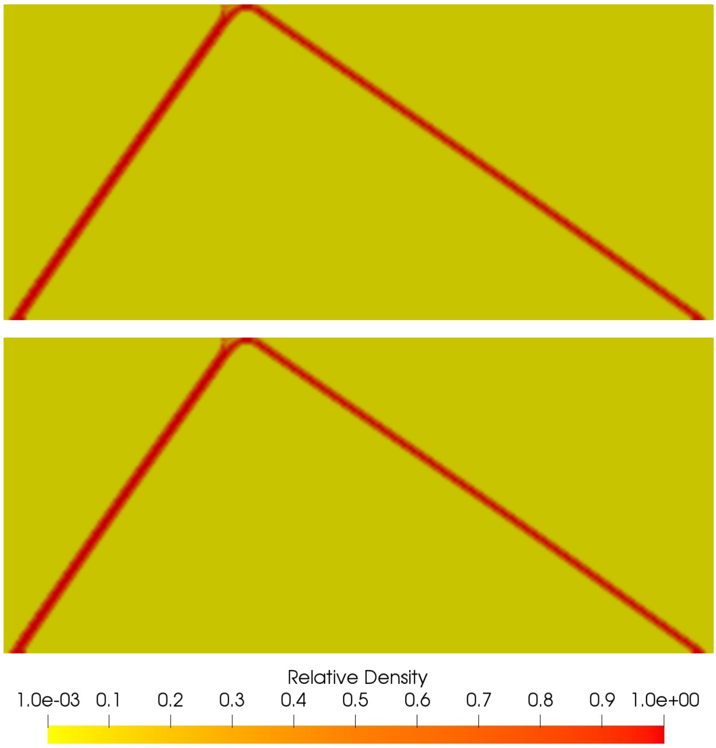


Figure 9.5. Beam with large height and upper load: Evolution of the relative density by using the Isogeometric Analysis [iterations 1200, 1500]

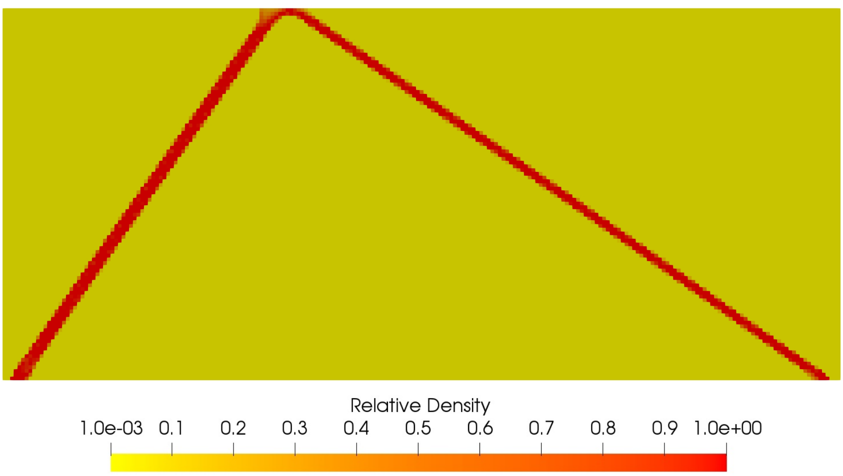


Figure 9.6. Beam with large height and upper load: Optimal solution by using the Finite Element Method

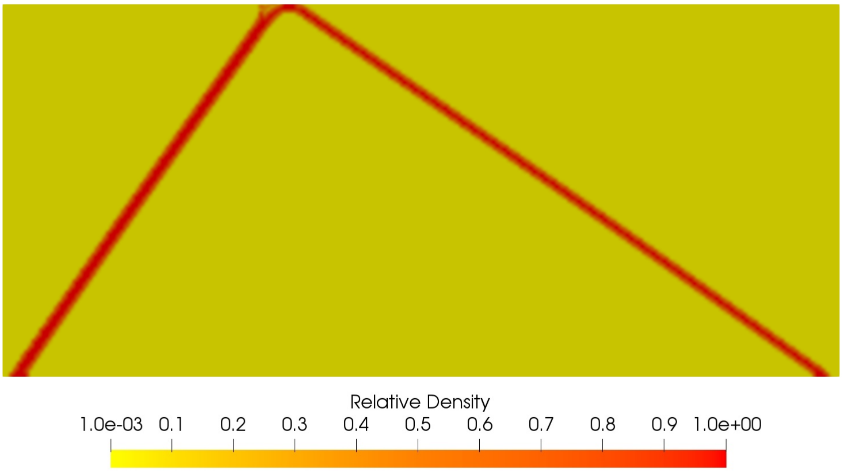


Figure 9.7. Beam with large height and upper load: Optimal solution by using the Isogeometric Analysis

As it can be seen in figures 9.6 and 9.7, the solution obtained with both methods coincides with the theoretical solution. Apart from that, the main difference between both solutions is due to the material layout definition of each method.

Figures 9.8 and 9.9 represent the stress state by means of the normalized stress of the solutions obtained with the conventional Finite Element Method and the Isogeometric Analysis. The normalized stress is obtained through the quotient between the stress in each point of the domain and the stress relaxation coefficient times the maximum allowable stress in this point.

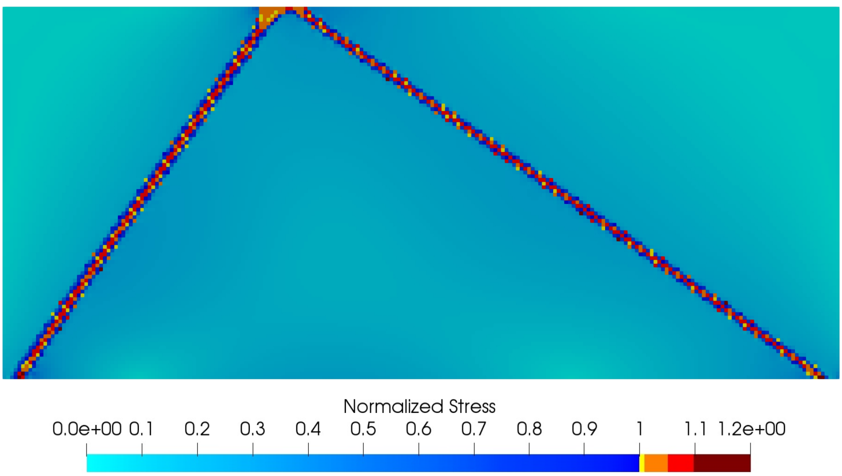


Figure 9.8. Beam with large height and upper load: Normalized stress by using the Finite Element Method

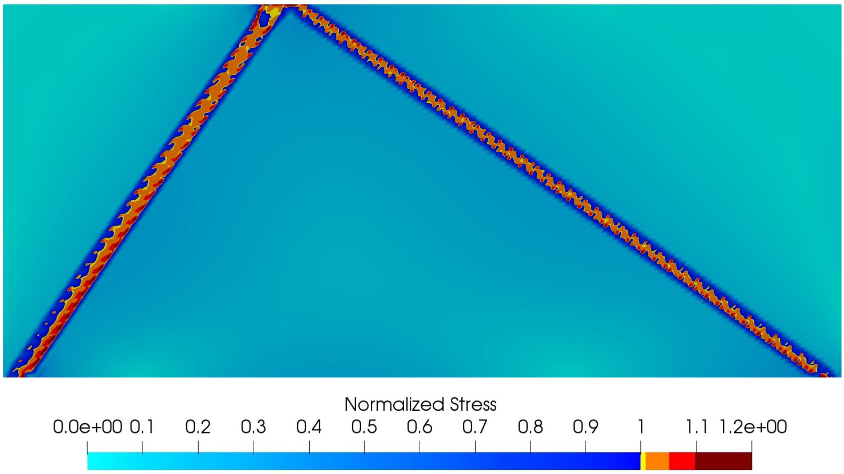


Figure 9.9. Beam with large height and upper load: Normalized stress by using the Isogeometric Analysis

Figures 9.8 and 9.9 show that there are regions whose stress value is slightly higher to their maximum allowable value. Nevertheless, this circumstance is due to the relaxation of the damage constraint, and tends to appear in the areas with relative density close to its lower limit since the own definition of the damage constraint implies that the stresses can take whatever value if the relative density is equal to its lower limit and the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint.

	Finite Element Method	Isogeometric Analysis
Number of design variables	22500	23154
Number of constraints	1	1
Number of iterations	1500	1500
Final weight (kg)	572.6	546.2
CPU time (h)	10.94	6.35
Time per iteration (s)	26.3	15.3
Structural analysis time / Total time	73.54%	31.09%
Sensitivity analysis time / Total time	24.90%	61.38%

Table 9.1. Beam with large height and upper load: General parameters of the problem

Tables 9.1, 9.2, 9.3 and 9.4 show the most important parameters used in the topology optimization problem. Regarding to the computational requirements, it is possible to observe that the CPU time required to solve the problem with the Isogeometric Analysis proposed is lower than with the conventional Finite Element Method.

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.2. Beam with large height and upper load: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-299	
2	0.001	300-599	
3	0.002	600-899	
4	0.003	900-1199	
5	0.005	1200-1500	

Table 9.3. Beam with large height and upper load: Value of the parameters during the resolution of the problem and range of application

	Finite Element Method	Isogeometric Analysis
Initial moving limits of the design variables	0.005	
Factor of evolution of the moving limits between sets of iterations	0.75	
Number of iterations between two consecutive modifications	150	

Table 9.4. Beam with large height and upper load: Evolution of the moving of the design variables during optimization

In the same way, the memory space required is small, basically just the needed to store the structural stiffness matrix in both methods due to the use of only one constraint to introduce the effect of local stress constraints. As a result of this, the conventional Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis. This circumstance has been commented in Chapter 8.

Furthermore, the number of iterations of the topology optimization algorithm used to obtain the final solution has been in both cases of 1500. However, it is possible to observe in figures 9.3 and 9.5 that there are no considerable differences between the solution obtained for a smaller number of iterations and the final solution. For this reason, it is possible to ensure that the optimal solution can be achieved with a smaller number of iterations.

Even though the ideal situation would be to end the loop due to the convergence of the solution, this convergence does not exist properly because of the high non-linearity of the problem and the use of first order Taylor expansions, especially in case of the Damage Constraint.

On the other hand, the final solutions obtained with both methods are equivalent in terms of topology, since their shape regarding to the material layout is the same. The final solution consists in two bars that connect the point where the external load is applied and the fixed supports.

Finally, it is important to remark that the difference regarding the amount of material required to manufacture the optimal solution obtained with both methods supposes only the 5 % of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout will not mean an important difference in the results obtained regarding to the structural weight, since this difference can be considered assumable in practice.

9.2.2. Cantilever beam

The second example of this section is a cantilever beam. This example will have a rigid support in the left border of the domain (null horizontal and vertical displacements). Moreover, a vertical load will be applied in the middle of the right border of the domain. This example has been also included in this section because of its extensive study in the topology optimization field. In the same way that for the previous example, it will be a test sample.

The dimensions of the domain used to solve this problem and the position of the external loads applied over the structure can be seen in figure 9.10. Furthermore, the structural weight will be considered as a structural load.

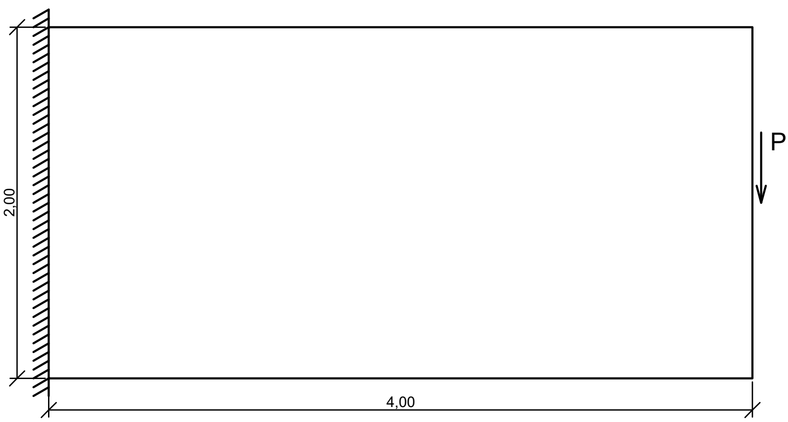


Figure 9.10. Cantilever beam: Problem statement. (units - m)

The structural domain will be discretized by means of a regular mesh of 200 x 100 = 20000 divisions of the domain. These divisions will be quadratic serendipity elements with 8 nodes in the case of the Finite Element Method and quadratic knot spans with 9 nodes in the case of the Isogeometric Analysis. On the other hand, the structural thickness will be in this case 0.18 m.

Even though the external load applied over the structure is a point load, it will be distributed over 12 adjacent elements in order to avoid the stress concentration phenomenon. The value of this point load will be equal to 40 10^3 kN. The material used for the design of this structure is steel with: material density $\gamma_{mat} = 7850\ kg/m^3$, Young's modulus $E = 2.1\ 10^5\ MPa$, Poisson's ratio $\nu = 0.3$ and yield stress $\tilde{\sigma}_{max} = 230\ MPa$.

Figures 9.11, 9.12, 9.13 and 9.14 represent the evolution of the optimal design during the topology optimization process, and figures 9.15 and 9.16 show the final solution obtained with the conventional Finite Element Method and the Isogeometric Analysis respectively.

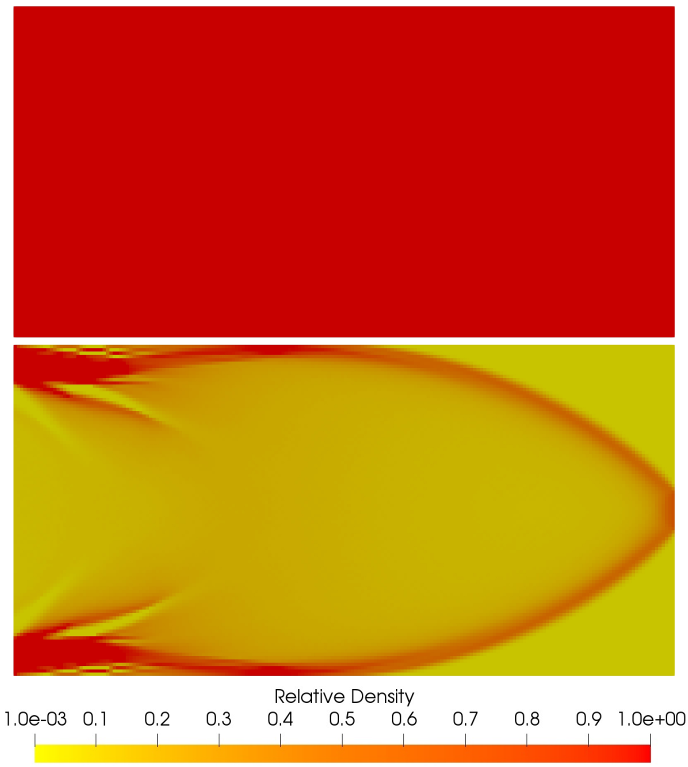


Figure 9.11. Cantilever beam: Evolution of the relative density by using the Finite Element Method [iterations 0, 2000]

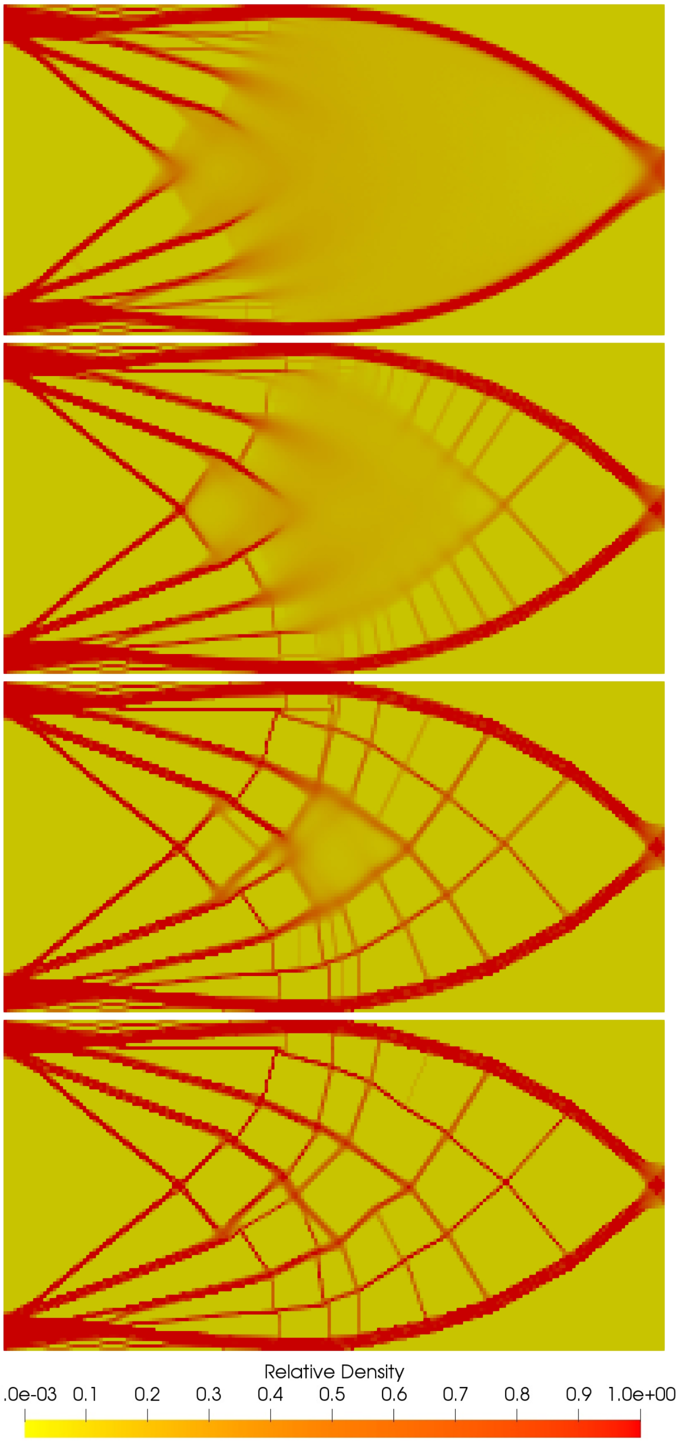


Figure 9.12. Cantilever beam: Evolution of the relative density by using the Finite Element Method [iterations 4000, 6000, 8000, 10000]

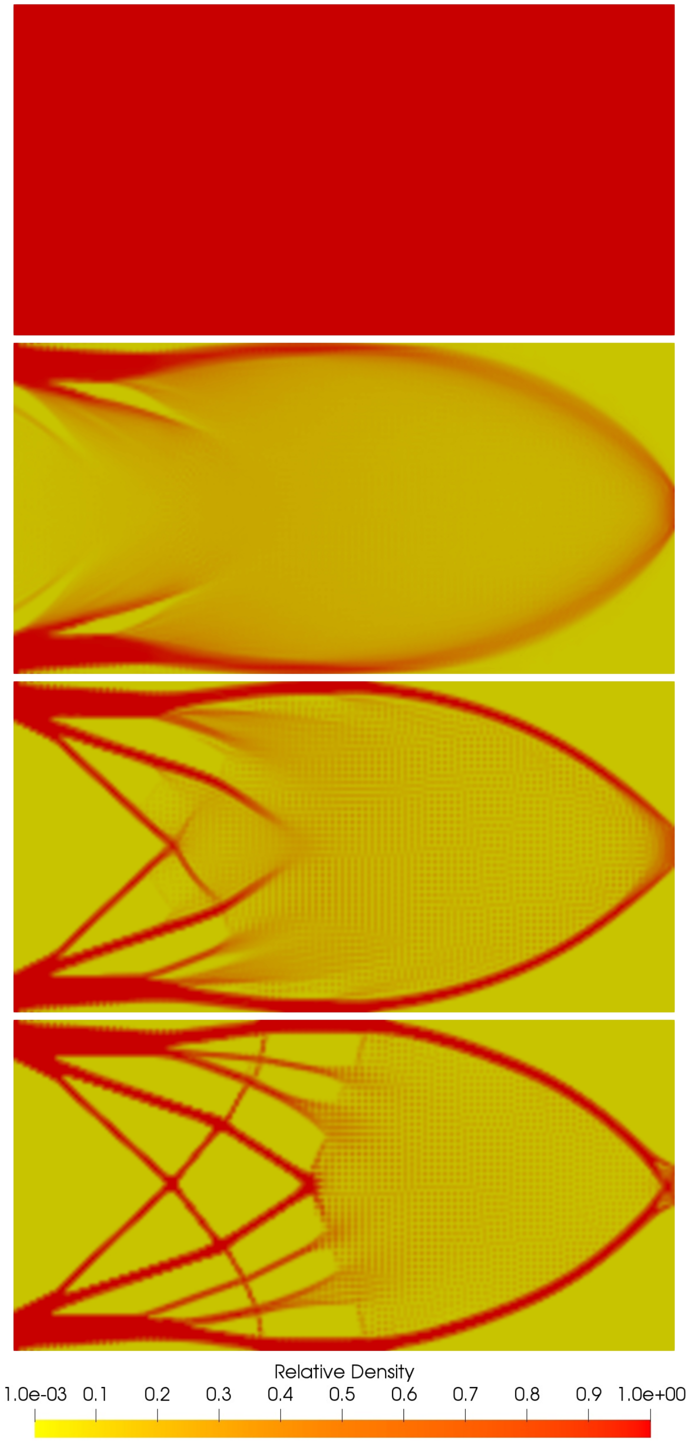


Figure 9.13. Cantilever beam: Evolution of the relative density by using the Isogeometric Analysis [iterations 0, 2600, 5200, 7800]

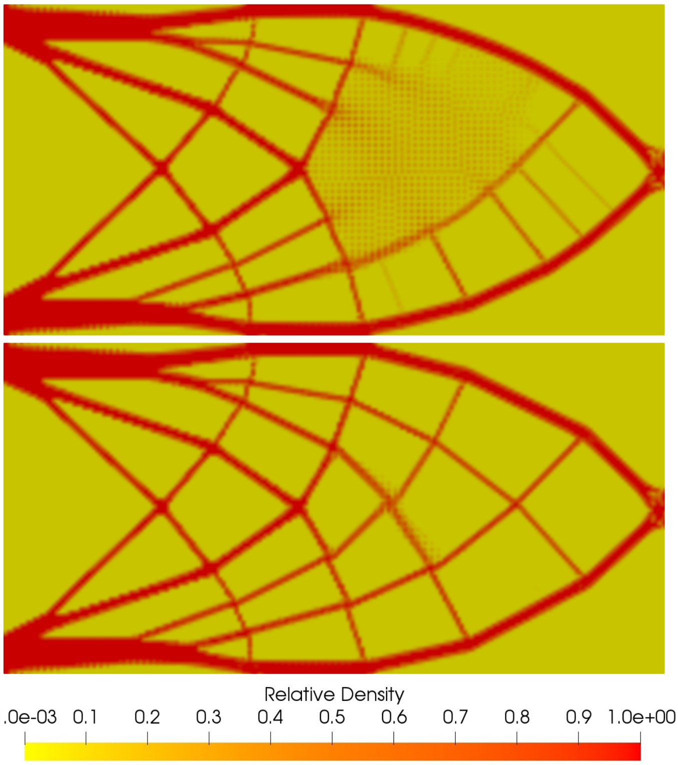


Figure 9.14. Cantilever beam: Evolution of the relative density by using the Isogeometric Analysis [iterations 10400, 13000]

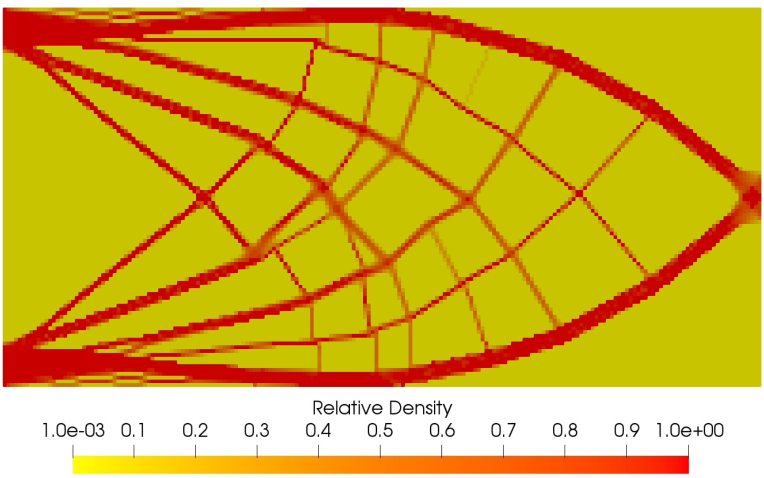


Figure 9.15. Cantilever beam: Optimal solution by using the Finite Element Method

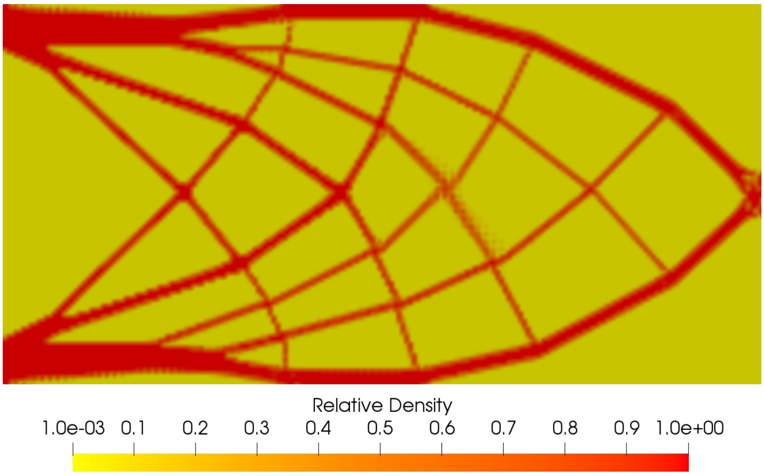


Figure 9.16. Cantilever beam: Optimal solution by using the Isogeometric Analysis

As it can be seen in figures 9.15 and 9.16, the solution obtained with both methods coincides with the solutions obtained previously. Apart from that, the main difference between both solutions is due to the way used to define the material layout with each method.

On the other hand, figures 9.17 and 9.18 represent the stress state by means of the normalized stress of the solutions obtained with the different formulations proposed in this thesis: the conventional Finite Element Method and the Isogeometric Analysis. The normalized stress is obtained through the quotient between the stress in each point of the domain and the stress relaxation coefficient times the maximum allowable stress in that point.

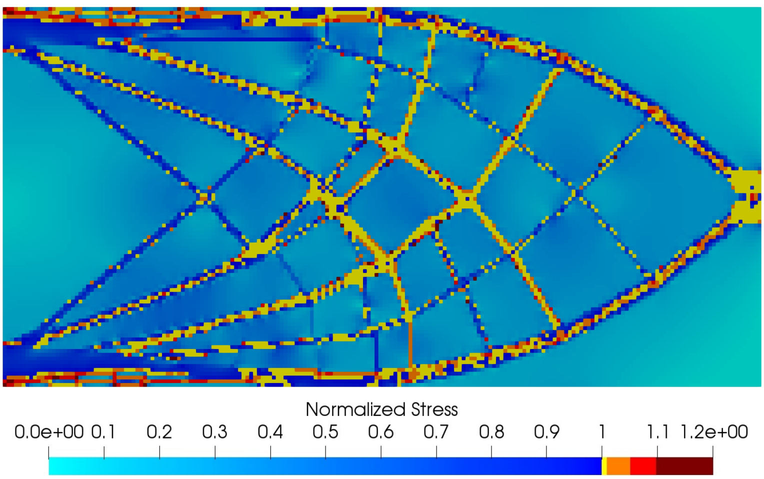


Figure 9.17. Cantilever beam: Normalized stress by using the Finite Element Method

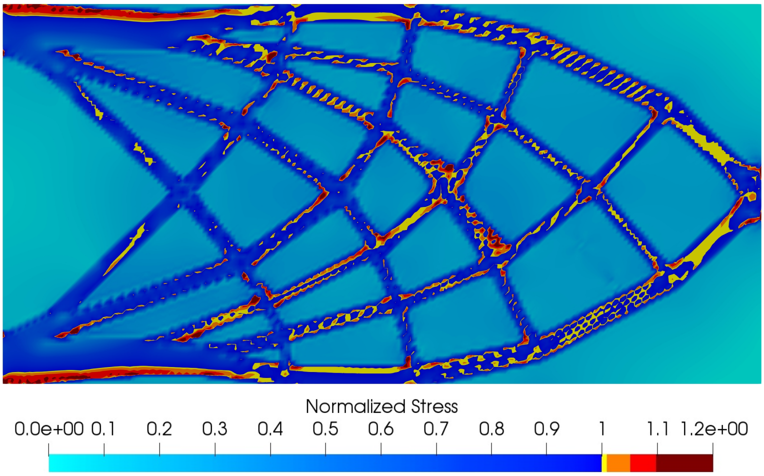


Figure 9.18. Cantilever beam: Normalized stress by using the Isogeometric Analysis

Figures 9.17 and 9.18 show that there are regions whose stress value is slightly higher to their maximum allowable value. Nonetheless, this circumstance is due to the relaxation of the damage constraint, and tends to appear in the areas with relative density close to its lower limit since the own definition of the damage constraint implies that the stresses can take whatever value if the relative density is equal to its lower limit and the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint.

Additionally, the most important characteristic of this example is the existence of symmetry in the final solution obtained between the upper and the lower part of the domain. This circumstance can be easily explained since the structural model used is linear and the material chosen is steel that has a similar behavior against traction and compression forces. That is, nonlinear effects like buckling or big displacements have not been considered.

Tables 9.5, 9.6, 9.7 and 9.8 show the value of the most important parameters of the problem. The CPU time required to solve the problem with the Isogeometric Analysis proposed is lower than with the conventional Finite Element Method despite of the larger number of iterations required.

The memory space required is basically the needed to store the structural stiffness matrix in both methods, since just one constraint is used to introduce the effect of local stress constraints. As a result of this, the Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis. This circumstance has been also commented in Chapter 8.

Moreover, the number of iterations used has been 10000 in case of the conventional Finite Element Method and 13000 in case of the Isogeometric Analysis. In contrast to the previous example, it is possible to observe in figures 9.12 and 9.14 that there are

	Finite Element Method	Isogeometric Analysis
Number of design variables	20000	20604
Number of constraints	1	1
Number of iterations	10000	13000
Final weight (kg)	2208.5	2239.8
CPU time (h)	63.26	57.02
Time per iteration (s)	22.8	15.8
Structural analysis time / Total time	74.34%	27.49%
Sensitivity analysis time / Total time	23.14%	63.57%

Table 9.5. Cantilever beam: General parameters of the problem

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.6. Cantilever beam: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-1999	0-2599
2	0.001	2000-3999	2600-5199
3	0.002	4000-5999	5200-7799
4	0.003	6000-7999	7800-10399
5	0.005	8000-10000	10400-13000

Table 9.7. Cantilever beam: Value of the parameters during the resolution of the problem and range of application

considerable differences between the solution obtained for an intermediate number of iterations and the final solution. Consequently, the number of iterations and the CPU time can not be reduced.

Although the ideal situation would be to stop the algorithm when convergence is achieved, it is not suitable in this case due to the high non-linearity of the problem, especially in the Damage Constraint.

On the other hand, the final solution obtained with both methods is equivalent in terms of topology, since their shape regarding to the material layout is approximately the same. The final solution of this problem consists in a certain number of bars that tend to coincide with the isostatic lines.

Nevertheless, it is possible that the number of bars and their position will be different depending on the method employed. This difference is related with the material layout

	Finite Element Method	Isogeometric Analysis
Initial moving limits of the design variables	0.005	
Factor of evolution of the moving limits between sets of iterations	0.75	
Number of iterations between two consecutive modifications	1000	1300

Table 9.8. Cantilever beam: Evolution of the moving of the design variables during optimization

discretization. In the conventional Finite Element Method, the value of the relative density in whatever point of each element is related with only one design variable, however, in the Isogeometric Analysis this value is a linear combination of design variables.

Thus, when the penalty coefficient is introduced in the formulation of the problem with the Isogeometric Analysis, the design variables whose value is close to their minimum allowable value tend to pull the nearest design variables to their minimum allowable value. This circumstance is due to the higher continuity order of this method in comparison with the conventional Finite Element Method what means that the transitions between the minimum and the maximum allowable value will not be immediate. Consequently, the softer bars of the conventional Finite Element Method, tend to disappear in the Isogeometric Analysis.

Finally, it is important to remark that the amount of material for the optimal solution is similar with both methods since the difference is only the 1.5% of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout will not suppose an important difference in the results obtained regarding to its structural weight.

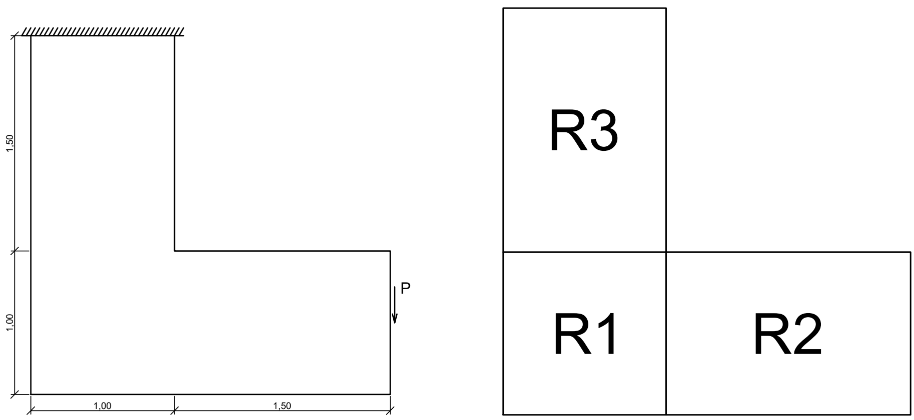
9.2.3. L-shaped beam

The last example of this section is a L-shaped beam. This example has a rigid support in the upper border of the domain, what means null horizontal and vertical displacements. Furthermore, a vertical load will be applied in the middle of the right border of the domain. This example has been also included in this section due to its extensive study in the topology optimization field. In this case, it is possible to test the Multiregion approach introduced in the Chapter 3 for the solution of problems with no rectangular domains by means of the Isogeometric Analysis since the domain used in the definition of this example is not rectangular. By the contrary, the solution of this kind of problems does not suppose a problem with the conventional Finite Element Method because of the lack of continuity between adjacent elements.

The dimensions of the domain used to solve this problem and the position of the external loads applied over the structure can be seen in figure 9.19a. Moreover, the structural weight will be considered as a structural load.

The structural domain will be discretized by means of a mesh of 19600 divisions of the domain. These divisions are quadratic serendipity elements with 8 nodes in the case of the Finite Element Method and quadratic knot spans with 9 nodes in the case of the Isogeometric Analysis. On the other hand, the structural thickness will be 0.85 m.

The domain will have to be divided in 3 regions as it can be seen in figure 9.19b. This is due to the limitations of the Isogeometric Analysis when non-rectangular domains are required. In other words, only square or rectangular regions can be directly discretized with the Isogeometric Analysis. Additionally, it will be required compatibility between these regions, in terms of its dimensions, because of the continuity of the structure in the domain with respect to the relative density, the displacements field and the stress value in case of the Isogeometric Analysis.



(a) Problem statement. (units - m)

(b) Division of the domain in rectangular regions

Figure 9.19. L-shaped beam: Problem statement and domains discretization

Although the external load applied over the structure is a point load, it will be distributed over 4 adjacent elements in order to avoid stress concentrations. The value of this point load will be 54 10^3 kN. The material used for the design of this structure is steel with: material density $\gamma_{mat} = 7850\ kg/m^3$, Young's modulus $E = 2.1\ 10^5\ MPa$, Poisson's ratio $\nu = 0.3$ and yield stress $\tilde{\sigma}_{max} = 230\ MPa$.

Figures 9.20 and 9.21 show the evolution of the optimal design during the topology optimization process, and figures 9.22 and 9.23 represent the final solution obtained with the formulations proposed in this thesis.

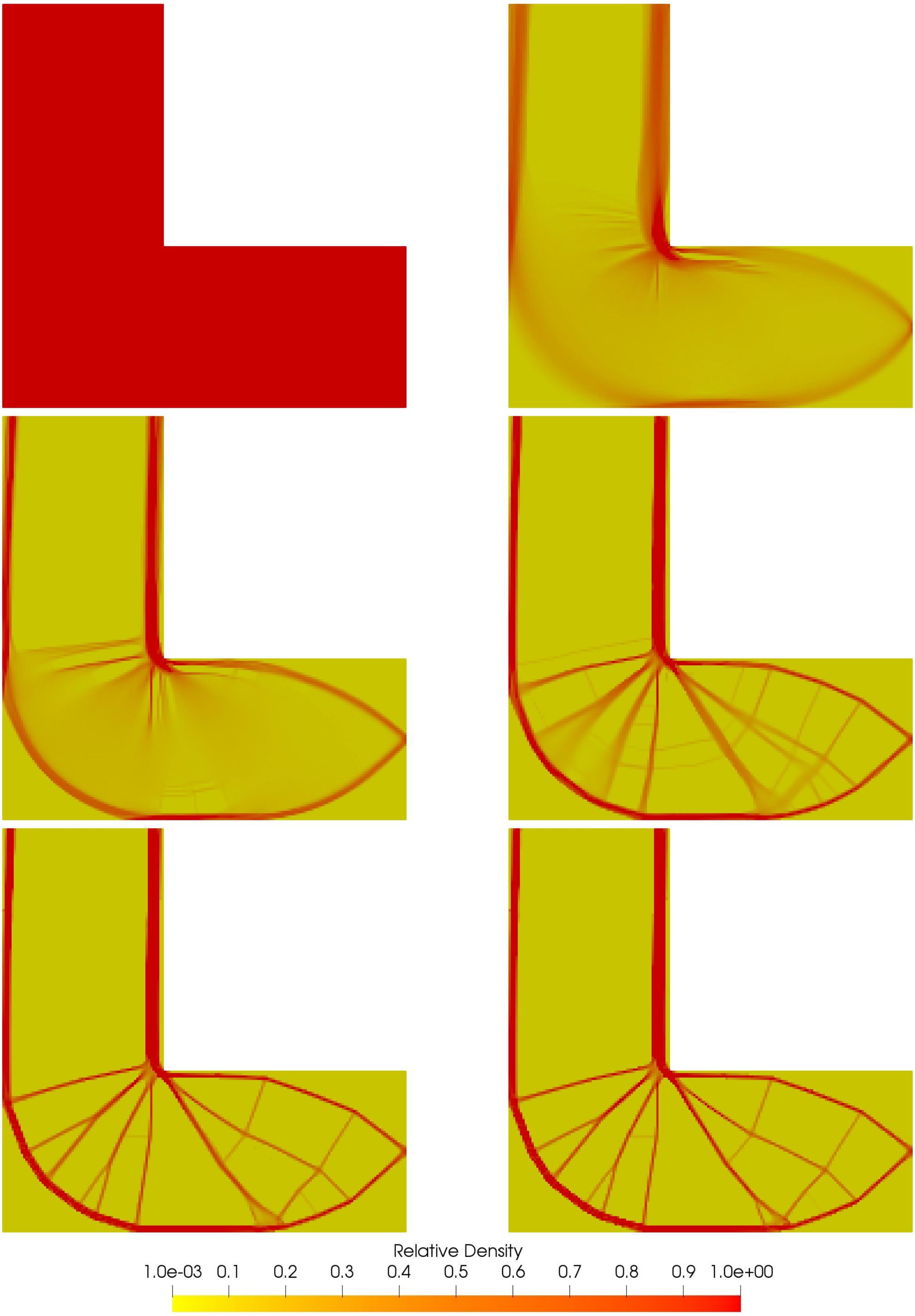


Figure 9.20. L-shaped beam: Evolution of the relative density by using the Finite Element Method [iterations 0, 600, 1200, 1800, 2400, 3000]

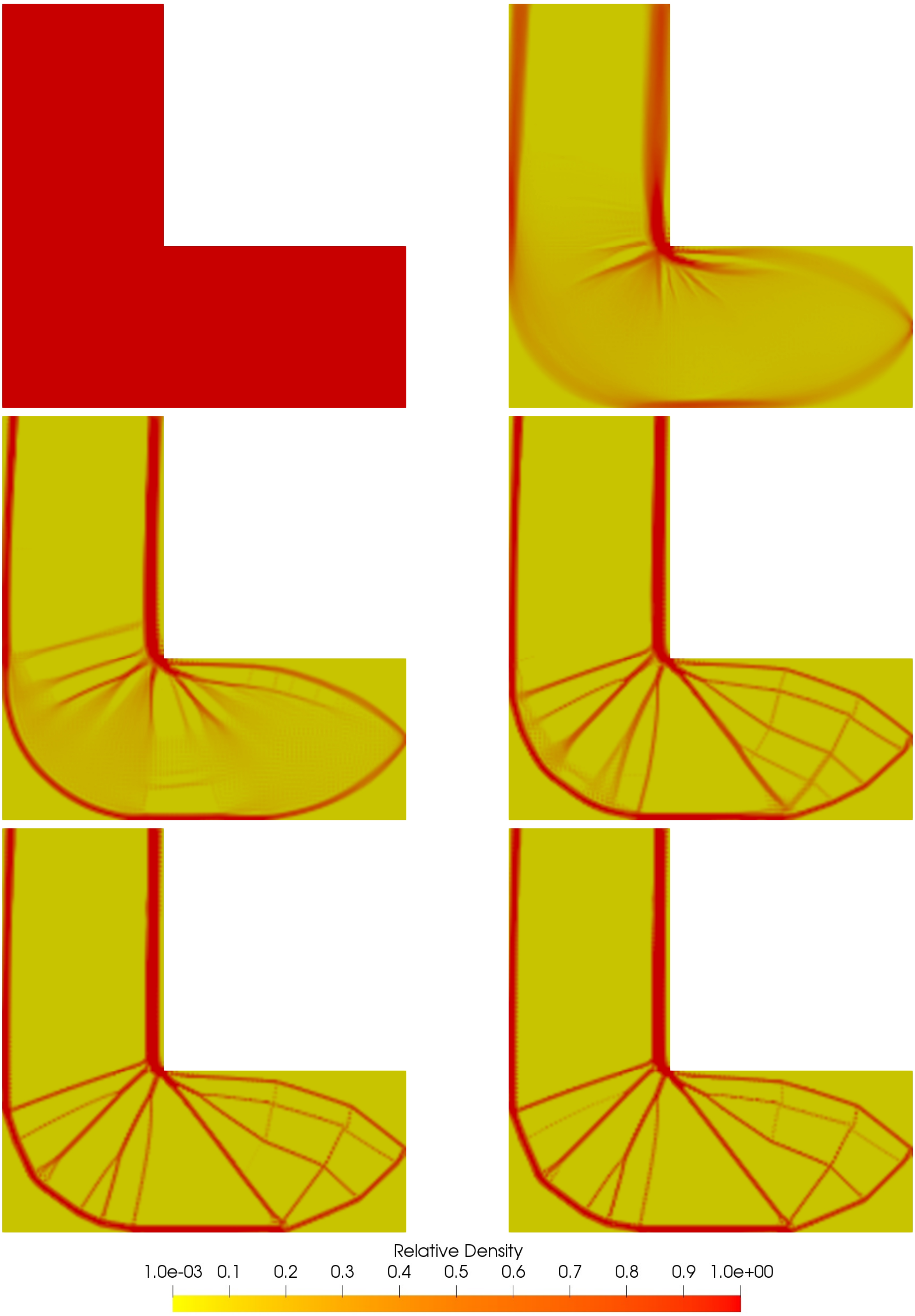


Figure 9.21. L-shaped beam: Evolution of the relative density by using the Isogeometric Analysis [iterations 0, 800, 1600, 2400, 3200, 4000]

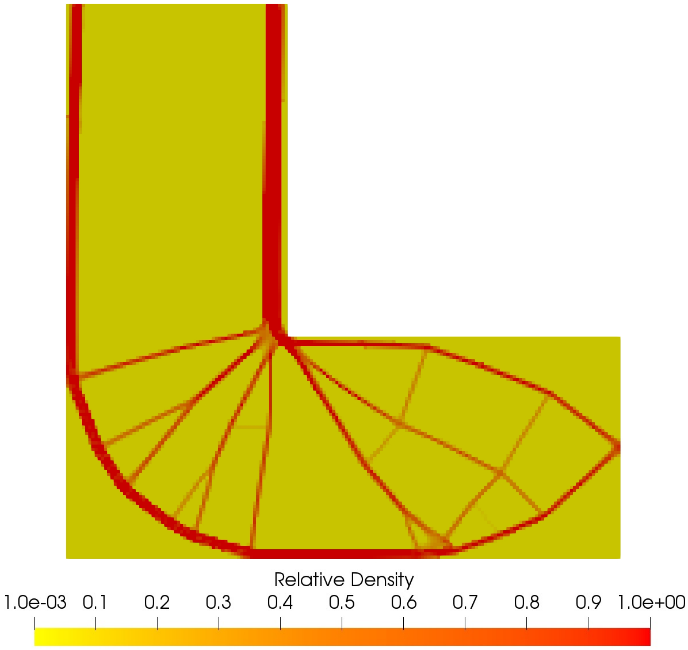


Figure 9.22. L-shaped beam: Optimal solution by using the Finite Element Method

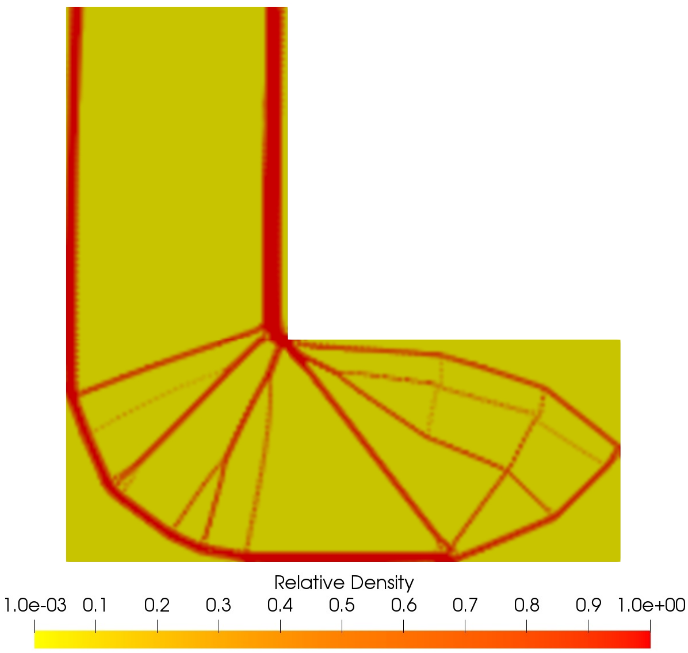


Figure 9.23. L-shaped beam: Optimal solution by using the Isogeometric Analysis

As it can be seen in figures 9.22 and 9.23, the solution obtained with both methods coincides with the solutions obtained previously for the same problem. Apart from that, the main difference between both solutions is due to the material layout definition with each method.

Conversely, figures 9.24 and 9.25 represent the stress state by means of the normalized stress of the solutions obtained with the different formulations. The value of the normalized stress is equal to the quotient between the stress in each point of the domain and the stress relaxation coefficient times the maximum allowable stress in that point.

Figures 9.24 and 9.25 show that there are regions whose stress value is slightly higher than their maximum allowable value. However, this circumstance is due to the relaxation of the damage constraint, and tends to appear in areas with relative density close to its lower limit since the own definition of the damage constraint implies that the stresses can take whatever value if the relative density is equal to its lower limit and the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint. Moreover, in this case, this circumstance also takes place in the proximities of the points of contact among three regions, in that the domain had been divided because of the stress concentration.

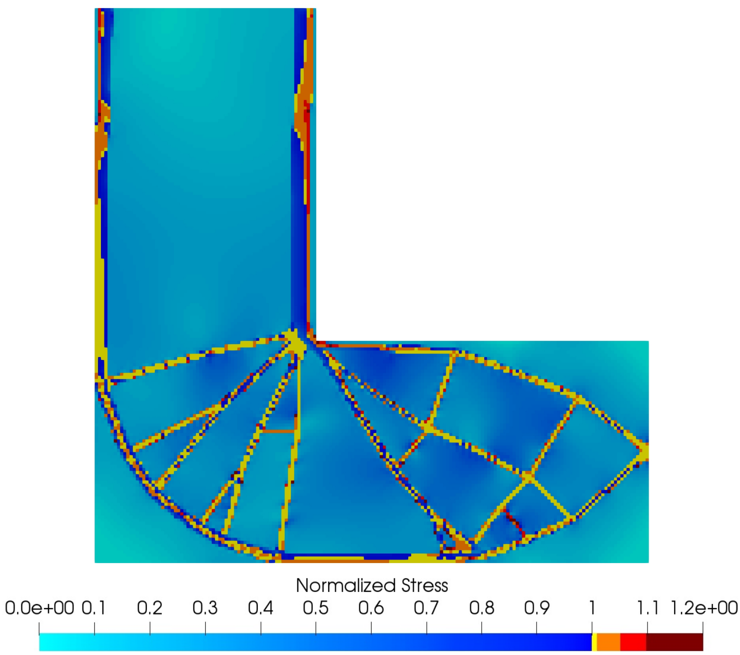


Figure 9.24. L-shaped beam: Normalized stress by using the Finite Element Method

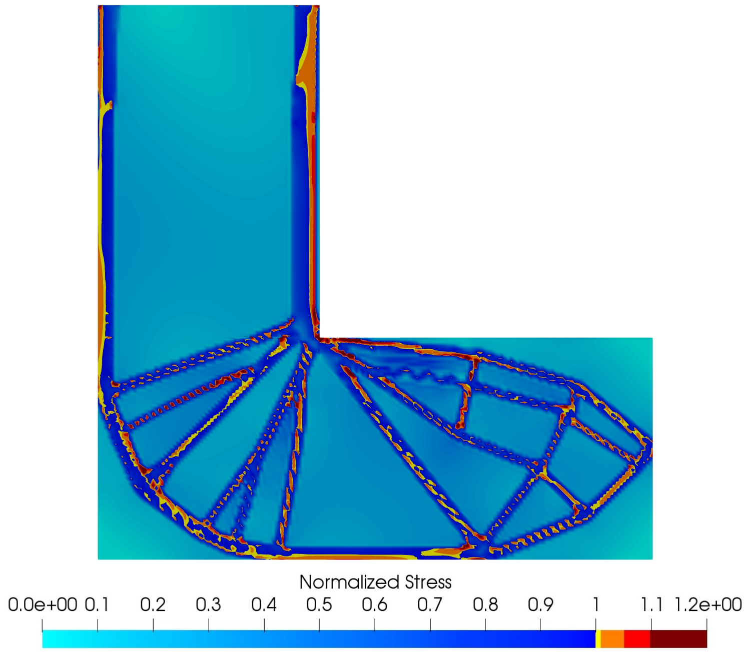


Figure 9.25. L-shaped beam: Normalized stress by using the Isogeometric Analysis

	Finite Element Method	Isogeometric Analysis
Number of design variables	19600	20448
Number of constraints	1	1
Number of iterations	3000	4000
Final weight (kg)	3135.6	3209.0
CPU time (h)	14.87	14.39
Time per iteration (s)	17.9	13.0
Structural analysis time / Total time	69.49%	23.69%
Sensitivity analysis time / Total time	27.81%	67.58%

Table 9.9. L-shaped beam: General parameters of the problem

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.10. L-shaped beam: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-599	0-799
2	0.001	600-1199	800-1599
3	0.002	1200-1799	1600-2399
4	0.003	1800-2399	2400-3199
5	0.005	2400-3000	3200-4000

Table 9.11. L-shaped beam: Value of the parameters during the resolution of the problem and range of application

<table>
<tr><th></th><th>Finite Element Method</th><th>Isogeometric Analysis</th></tr>
<tr><td>Initial moving limits of the design variables</td><td colspan="2">0.005</td></tr>
<tr><td>Factor of evolution of the moving limits between sets of iterations</td><td colspan="2">0.75</td></tr>
<tr><td>Number of iterations between two consecutive modifications</td><td>300</td><td>400</td></tr>
</table>

Table 9.12. L-shaped beam: Evolution of the moving of the design variables during optimization

Tables 9.9, 9.10, 9.11 and 9.12 show the most important parameters of the problem. It is possible to observe that the CPU time required to solve the problem with the Isogeometric Analysis is lower than with the conventional Finite Element Method despite of having to use more iterations to obtain the solution with the Isogeometric Analysis.

The memory space required is essentially the required to store the structural stiffness matrix in both methods since only one constraint is used. As a result of this, the conventional Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis.

On the other side, the number of iterations used to obtain the final solution has been 3000 in case of the conventional Finite Element Method and 4000 in case of the Isogeometric Analysis. However, it is possible to observe in figures 9.20 and 9.21 that there are no considerable differences between the solution obtained for a smaller number of iterations and the final solution. This circumstance allows to ensure that the solution has converged to its optimum.

Although the ideal situation would be to stop the algorithm with the convergence of the solution, it is not suitable in this case due to the high non-linearity of the problem, especially in the Damage Constraint.

On the other hand, the final solutions obtained with both methods are equivalent in terms of topology, since their material layout is quite similar. The possible differences between both solutions are related to the material layout discretization. The final solution consists in two vertical bars that connect the fixed supports with the lower part of the domain in the region number 3, and arc with several radius that connect the arc with the point of contact between the three regions defined in the region number 1, and bars that coincides with the isostatic lines and connects the point where the load is applied with the structure described previously in the region number 2.

Finally, it is important to remark that the difference regarding the amount of material required to manufacture the optimal solution obtained with both methods is quite small since the difference is only the 2.5% of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout does not suppose an important difference in the results obtained with respect to its structural weight.

9.3. Two-dimensional examples

Previous examples correspond to benchmark cases usually solved in topology optimization to test new approaches and models. Then, several two-dimensional examples of application in the civil and the aeronautical engineering field will be solved in this section: a beam with large height and load on the lower edge, an undercarriage, a mast and a MBB beam.

9.3.1. Beam with large height and load on the lower edge

The first example of this section is a beam with large height. This example will have fixed supports in the lower corners that is horizontal and vertical displacements suppressed on both sides. Moreover, a vertical distributed load will be applied in the middle of the span.

At this point, it is important to remark that it would be possible to calculate only one half of the structure since the structural problem is symmetric. In this case, the right half of the structure will be analyzed. As a result of this circumstance it will be necessary to modify the structural supports. It will be necessary to introduce the horizontal displacements constraint in the symmetry axis and the vertical displacements are not constrained.

First, figure 9.26 shows a graphical scheme with the dimensions of the entire structure, the part of the structure that will be optimized and the position of the external loads applied over the structure. Moreover, it is important to establish that the structural weight will be considered as a structural load.

The structural domain considered will be discretized by means of a regular mesh of 132 x 120 = 15840 divisions of the domain. These divisions will be quadratic serendipity elements with 8 nodes in the case of the Finite Element Method and quadratic knot spans with 9 nodes in the case of the Isogeometric Analysis. On the other hand, the structural thickness will be in this case 0.30 m.

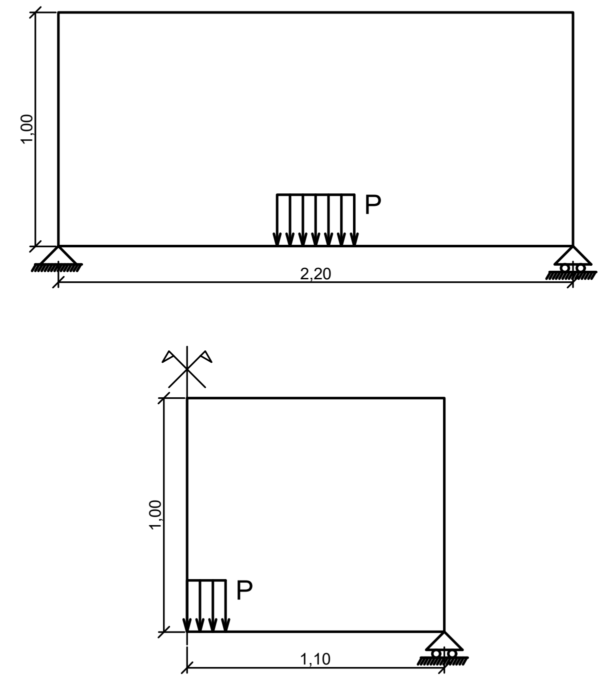


Figure 9.26. Beam with large height and load on the lower edge: Problem statement. (units - m)

An external distributed load of 240 10^3 kN/m is applied over 6 adjacent elements in the inferior part of the domain in the proximity of the symmetry axis. In the same way that in the previous examples, the material used for the design of this structure will be steel with: material density $\gamma_{mat} = 7850\ kg/m^3$, Young's modulus $E = 2.1\ 10^5$ MPa, Poisson's ratio $\nu = 0.3$ and yield stress $\tilde{\sigma}_{max} = 230\ MPa$.

Figures 9.27 and 9.28 represent the evolution of the optimal design during the topology optimization process, and figures 9.29 and 9.30 show the final solution obtained with the formulations proposed in this thesis.

As it can be seen in figures 9.29 and 9.30, the solutions obtained with both methods are practically equal. The main differences between both solutions correspond to the material layout discretization of each method. At this point, it is important to remark that the real solution of the problem proposed can be obtained by means of the symmetrical replication of the solution shown in figures 9.29 and 9.30 through the symmetry axis. Figures 9.31 and 9.32 represent the stress state by means of the normalized stress of the solutions obtained with the different formulations proposed in this thesis.

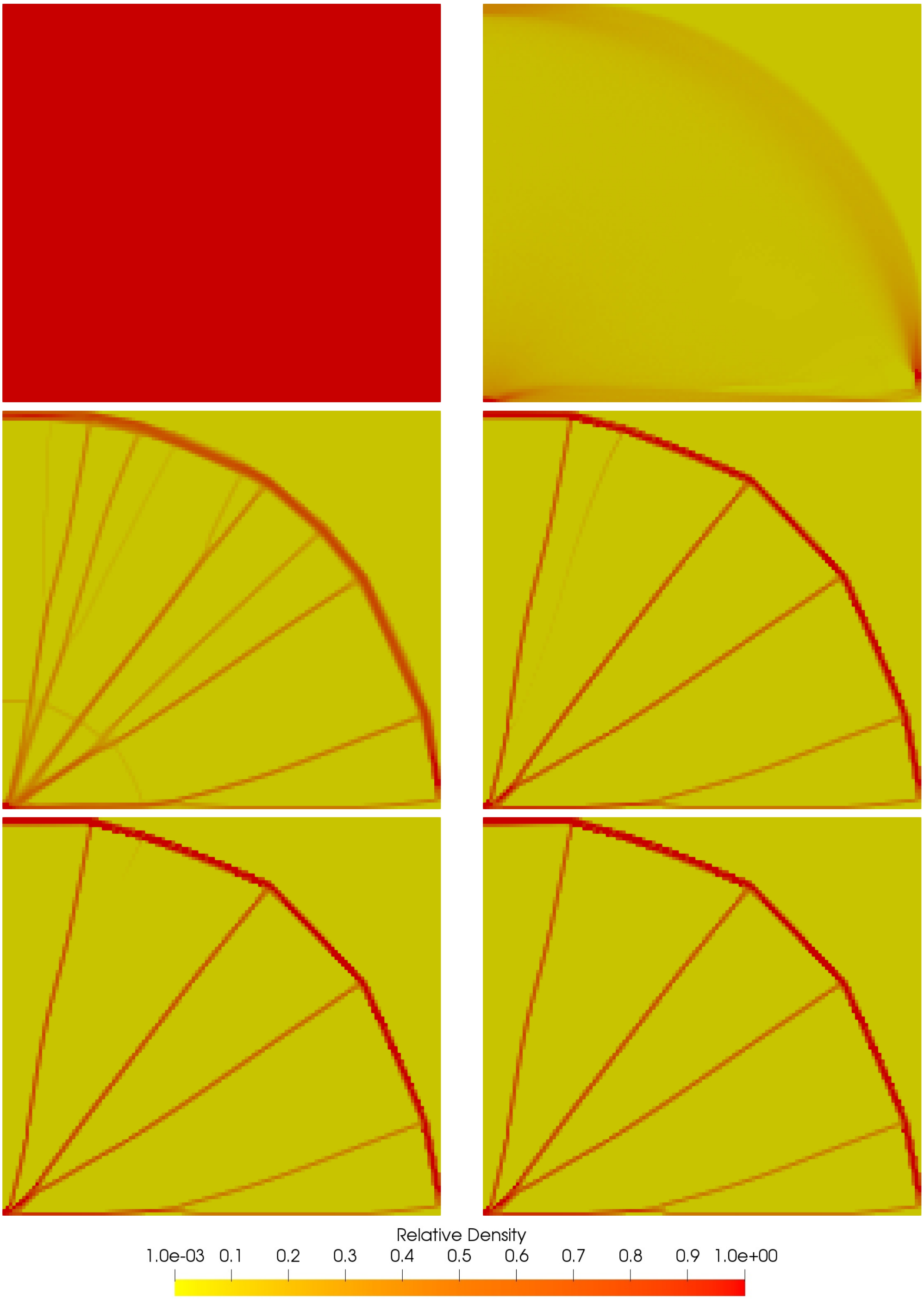


Figure 9.27. Beam with large height and load on the lower edge: Evolution of the relative density by using the Finite Element Method [iterations 0, 600, 1200, 1800, 2400, 3000]

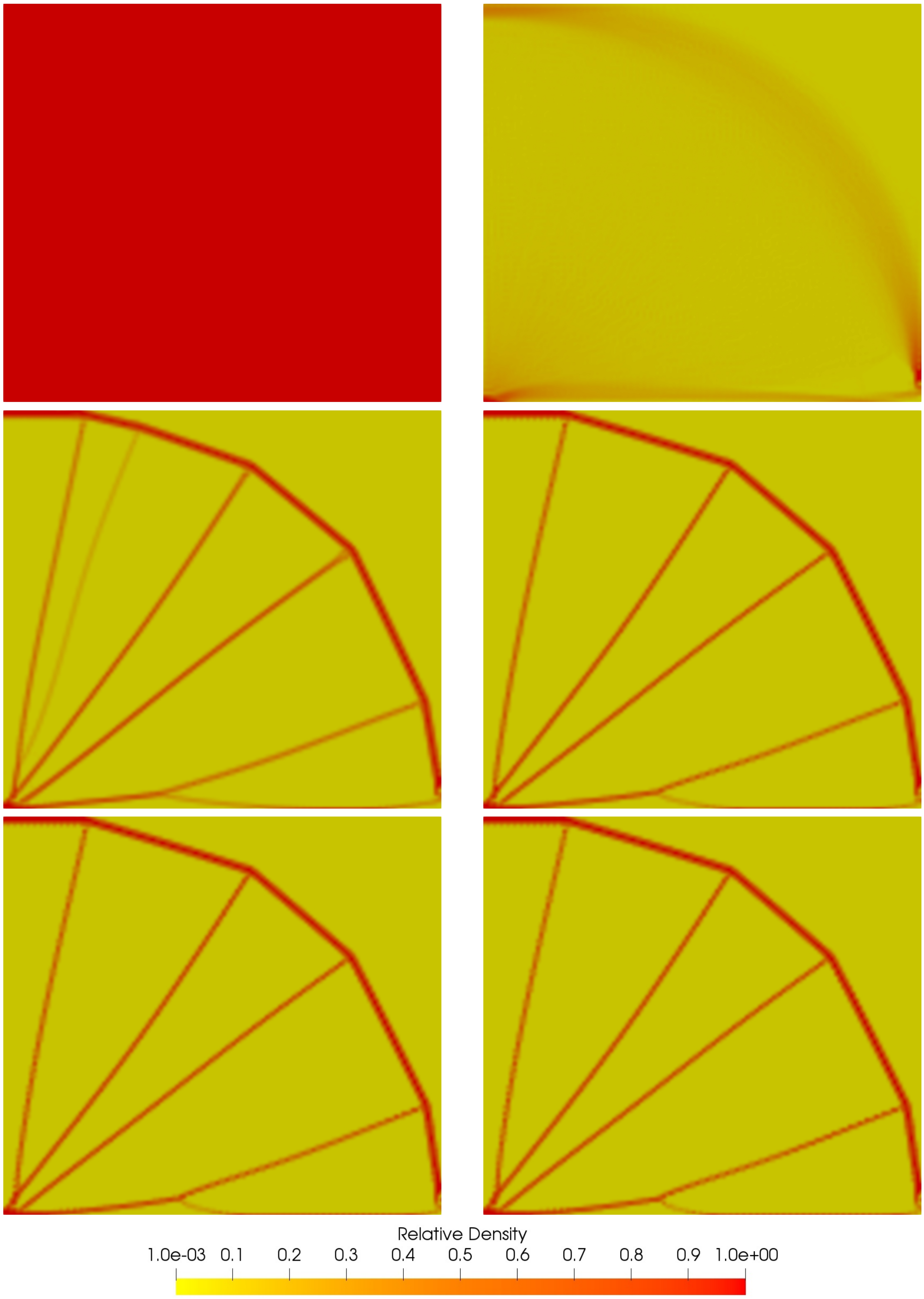


Figure 9.28. Beam with large height and load on the lower edge: Evolution of the relative density by using the Isogeometric Analysis [iterations 0, 700, 1400, 2100, 2800, 3500]

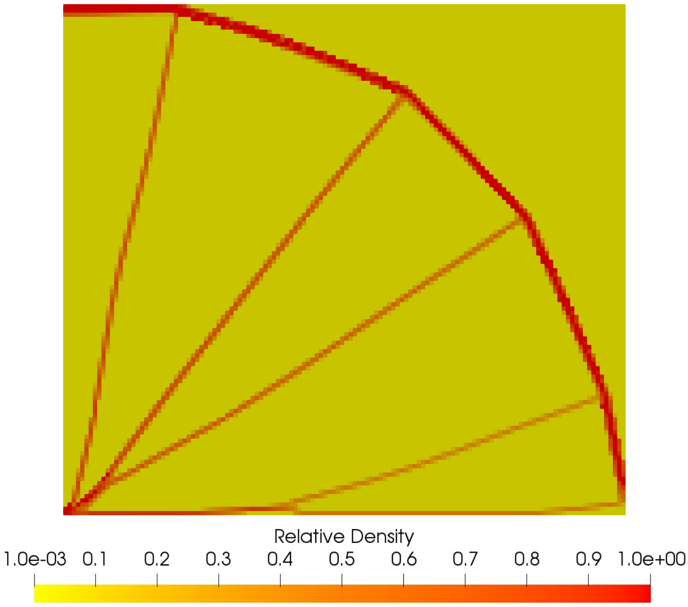


Figure 9.29. Beam with large height and load on the lower edge: Optimal solution by using the Finite Element Method

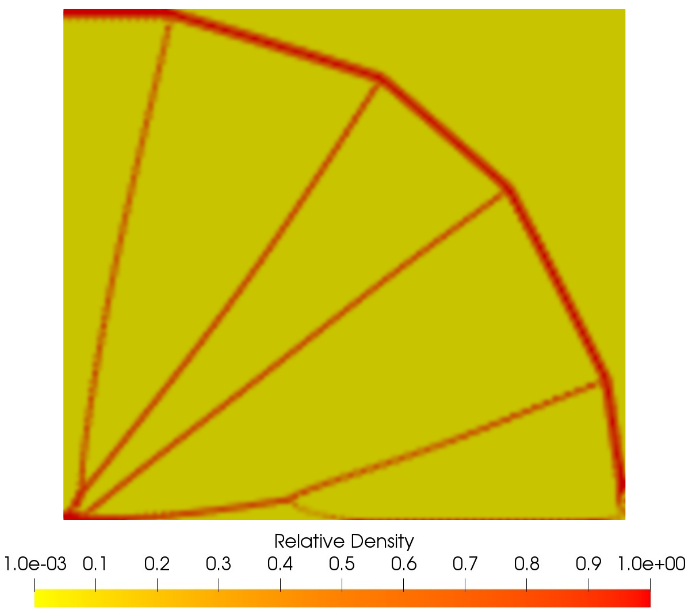


Figure 9.30. Beam with large height and load on the lower edge: Optimal solution by using the Isogeometric Analysis

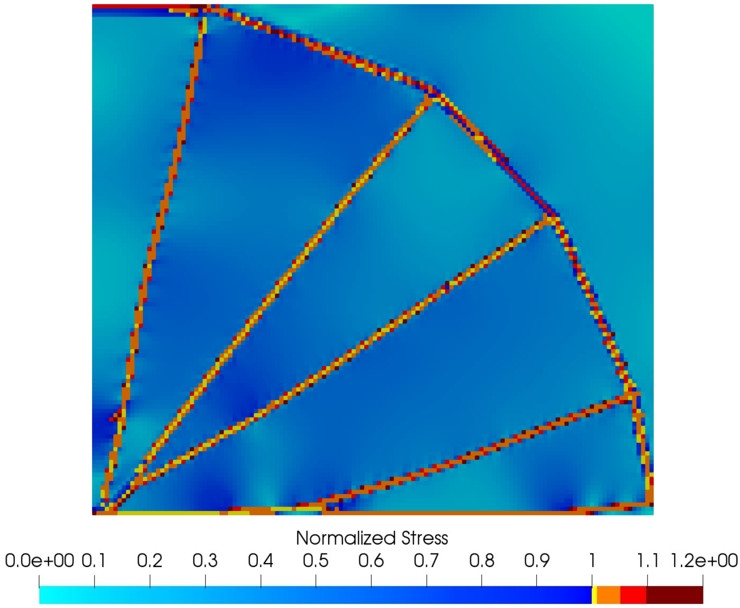


Figure 9.31. Beam with large height and load on the lower edge: Normalized stress by using the Finite Element Method

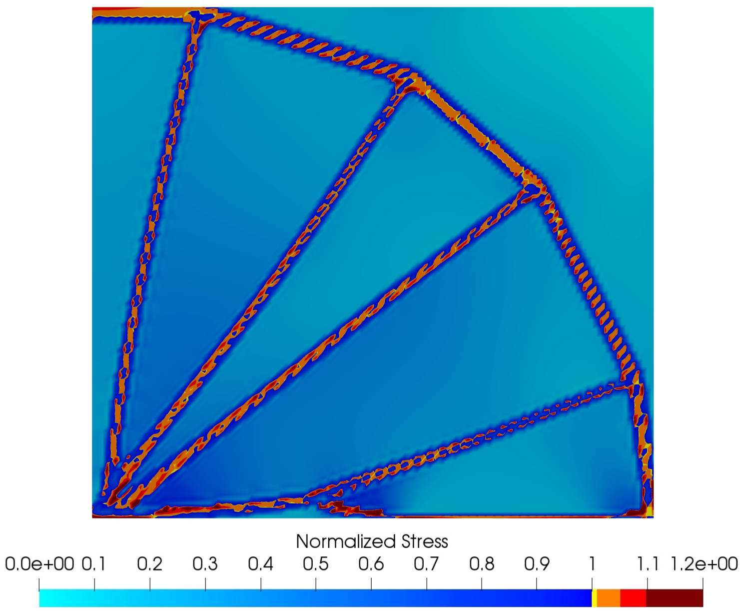


Figure 9.32. Beam with large height and load on the lower edge: Normalized stress by using the Isogeometric Analysis

Figures 9.31 and 9.32 show that there are regions that slightly exceed their maximum allowable value. However, this circumstance is due to the relaxation of the damage constraint, and tends to appear in areas with relative density close to its lower limit since the own definition of the damage constraint supposes that the stresses can take whatever value if the relative density is equal to its lower limit and, on the other hand, the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint.

	Finite Element Method	Isogeometric Analysis
Number of design variables	15840	16348
Number of constraints	1	1
Number of iterations	3000	3500
Final weight (kg)	136.3	139.8
CPU time (h)	19.12	10.14
Time per iteration (s)	23.0	10.5
Structural analysis time / Total time	83.63%	43.60%
Sensitivity analysis time / Total time	15.17%	49.02%

Table 9.13. Beam with large height and load on the lower edge: General parameters of the problem

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.14. Beam with large height and load on the lower edge: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-599	0-699
2	0.001	600-1199	700-1399
3	0.002	1200-1799	1400-2099
4	0.003	1800-2399	2100-2799
5	0.005	2400-3000	2800-3500

Table 9.15. Beam with large height and load on the lower edge: Value of the parameters during the resolution of the problem and range of application

	Finite Element Method	Isogeometric Analysis
Initial moving limits of the design variables	0.005	
Factor of evolution of the moving limits between sets of iterations	0.75	
Number of iterations between two consecutive modifications	300	350

Table 9.16. Beam with large height and load on the lower edge: Evolution of the moving of the design variables during optimization

Tables 9.13, 9.14, 9.15 and 9.16 show the value of the most important parameters of the problem. It is possible to observe that the CPU time required to solve the problem with the Isogeometric Analysis proposed is lower than with the conventional Finite Element Method despite of using more iterations to obtain the solution in case of the Isogeometric Analysis.

In the same way, the memory space required is small, basically just the needed to store the structural stiffness matrix in both methods due to the use of only one constraint to introduce the effect of local stress constraints. As a result of this, the conventional Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis formulation. This circumstance is due to the lower number of points required in the solution of the structural analysis what is related with the size of the global stiffness matrix and a lower bandwidth of the global stiffness matrix in case of the Isogeometric Analysis.

Moreover, the topology optimization algorithm used to obtain the final solution required 3000 iterations in the case of the conventional Finite Element Method and 3500 in the case of the Isogeometric Analysis. Nevertheless, it is possible to observe in figures 9.27 and 9.28 that there are few differences between the solution obtained for a smaller number of iterations and the final solution. This circumstance allows to ensure that the solution has converged to its optimum.

Although the ideal situation would be to stop the algorithm due to the convergence of the solution, this convergence does not exist properly due to the high non-linearity of the problem, especially in case of the Damage Constraint, since the approximation used in the solution of the problem for the Damage Constraint is linear.

On the other hand, the final solution obtained with both methods is equivalent in terms of topology, since the global material layout is equivalent. The final solution consists in a set of bars that represent a circumference arc and several radius of this circumference, being the load applied in the center of the circumference.

Nonetheless, it is possible that the number of bars and their positions will be different depending on the method employed what has influence in the material layout discretization. In the conventional Finite Element Method, the value of the relative density at each point of an element is directly related with only one design variable, nevertheless, in the Isogeometric Analysis this relationship is more complex, since this value is a combination of design variables.

As a result of this circumstance, when the penalty coefficient is introduced in the formulation of the problem with the Isogeometric Analysis, the design variables whose value is close to their minimum allowable value tend to pull the nearest design variables to their minimum allowable value. Consequently, the Isogeometric Analysis can not develop sharp transition from low densities to high densities.

Finally, it is important to remark that the difference regarding the amount of material required to manufacture the optimal solution obtained with both methods is quite small since the difference is only the 3% of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout will not mean an important difference in the results obtained with respect to its structural weight.

9.3.2. Undercarriage

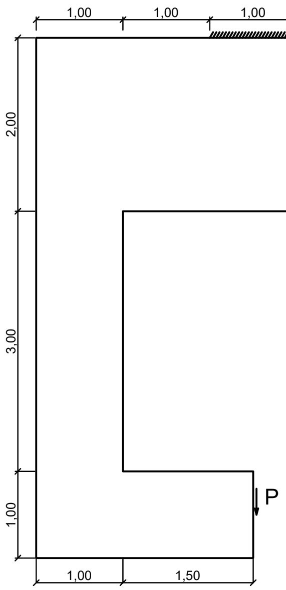
The second example of this section is an undercarriage. This example will have a rigid support in the right third of the upper border of the domain what means null horizontal and vertical displacements in this border. Moreover, a vertical load will be applied in the middle of the right border of the domain in its lower part.

The dimensions of the domain used to solve this problem and the position of the external loads applied over the structure can be seen in figure 9.33a. Furthermore, the structural weight will be considered as a structural load.

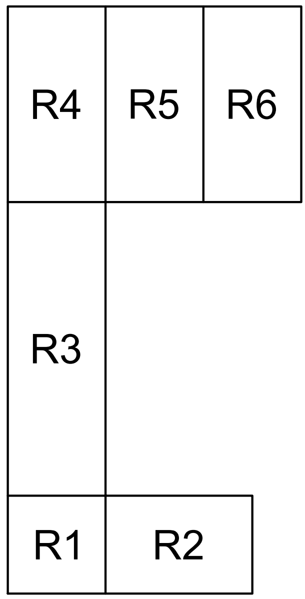
The structural domain will be discretized by means of a mesh of 18400 quadratic serendipity elements with 8 nodes in the case of the conventional Finite Element Method and the same number of quadratic knot spans with 9 nodes in the case of the Isogeometric Analysis. On the other hand, the structural thickness will be in this case 0.22 m.

At this point, it is important to remark that the domain will have to be divided in 6 regions (figure 9.33b). This is due to the limitations of the Isogeometric Analysis when non-rectangular domains are required. In other words, only square or rectangular regions can be directly discretized with the Isogeometric Analysis. Additionally, it will be required a level of continuity between two adjacent regions in terms of the relative density, the structural displacements and the stress value in case of the Isogeometric Analysis, for this reason, the borders that represent the contact between adjacent regions will have the same magnitude.

Although the external load applied over the structure is a point load, it will be distributed over 8 adjacent elements in order to avoid the stress concentration phenomenon. The point load will be 17 10^3 kN. The material used for the design of this structure is steel with: material density $\gamma_{mat} = 7850\ kg/m^3$, Young's modulus $E = 2.1\ 10^5\ MPa$, Poisson's ratio $\nu = 0.3$ and yield stress $\tilde{\sigma}_{max} = 230\ MPa$.



(a) Problem statement. (units - m)



(b) Division of the domain in rectangular regions

Figure 9.33. Undercarriage: Problem statement and domains discretization

Figures 9.34, 9.35, 9.36 and 9.37 represent the evolution of the optimal design during the topology optimization process, and figures 9.38 and 9.39 show the final solution obtained with the conventional Finite Element Method and the Isogeometric Analysis respectively.

As it can be seen in figures 9.38 and 9.39, the solution obtained with both methods coincides with the solutions obtained previously in the literature for the same problem. Apart from that, the main difference between both solutions is due to the material layout definition with each method.

On the other side, figures 9.40 and 9.41 represent the stress state by means of the normalized stress of the solutions obtained with the different formulations proposed in this thesis. The normalized stress is obtained through the quotient between the stress in each point of the domain and the stress relaxation coefficient times the maximum allowable stress in that point.

In the same way that for the previous examples, figures 9.40 and 9.41 show that there are regions whose stress value is slightly higher than their maximum allowable value. Nonetheless, this circumstance is due to the relaxation of the damage constraint, and tends to appear in areas with relative density close to its lower limit since the own definition of the damage constraint implies that the stresses can take whatever value if the relative density is equal to its lower limit and the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint. Moreover, in this case, this situation also takes place in the proximities of the points of contact among three regions, in that the domain had been divided because of the stress concentration.

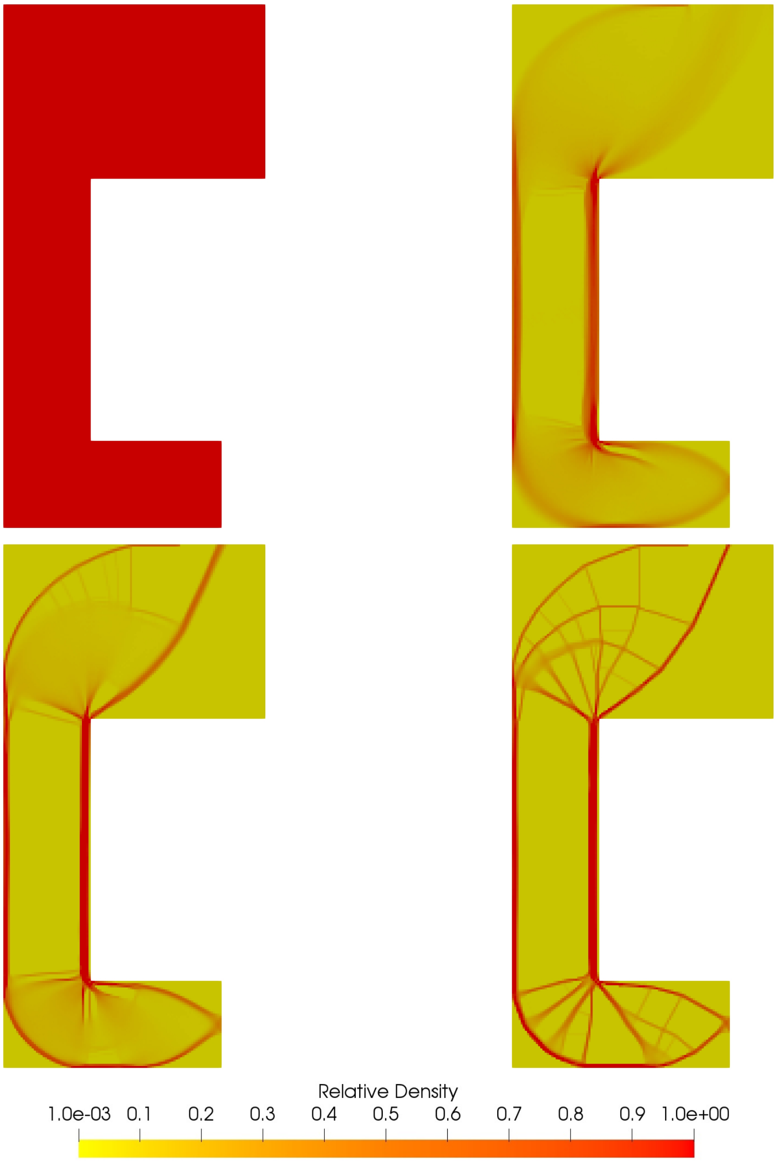


Figure 9.34. Undercarriage: Evolution of the relative density by using the Finite Element Method [iterations 0, 700, 1400, 2100]

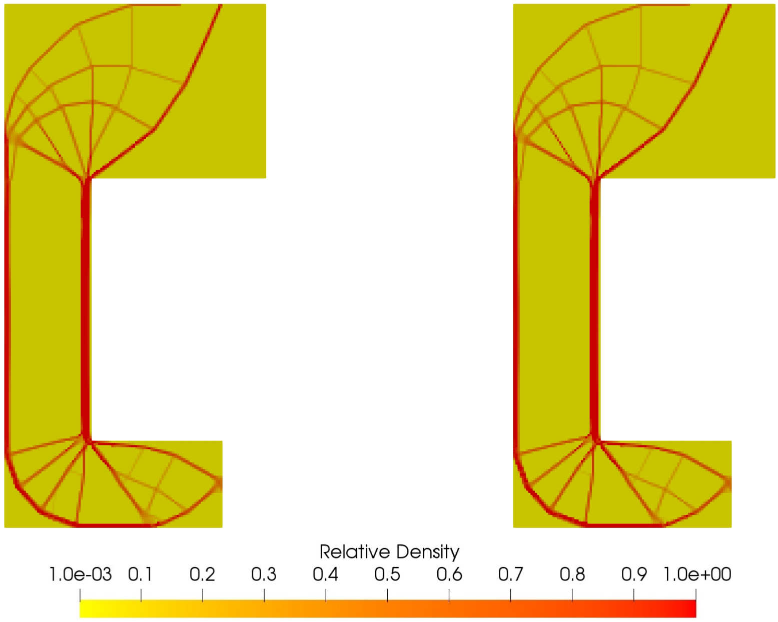


Figure 9.35. Undercarriage: Evolution of the relative density by using the Finite Element Method [iterations 2800, 3500]

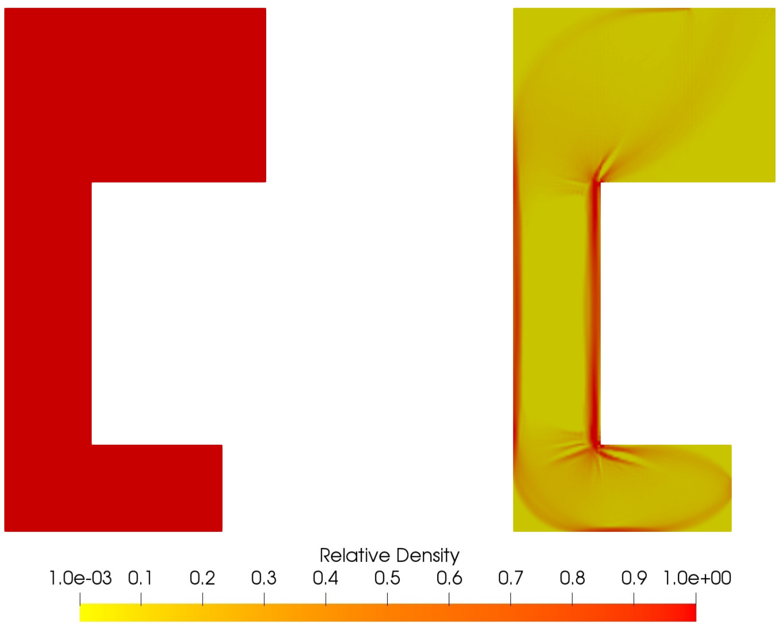


Figure 9.36. Undercarriage: Evolution of the relative density by using the Isogeometric Analysis [iterations 0, 700]

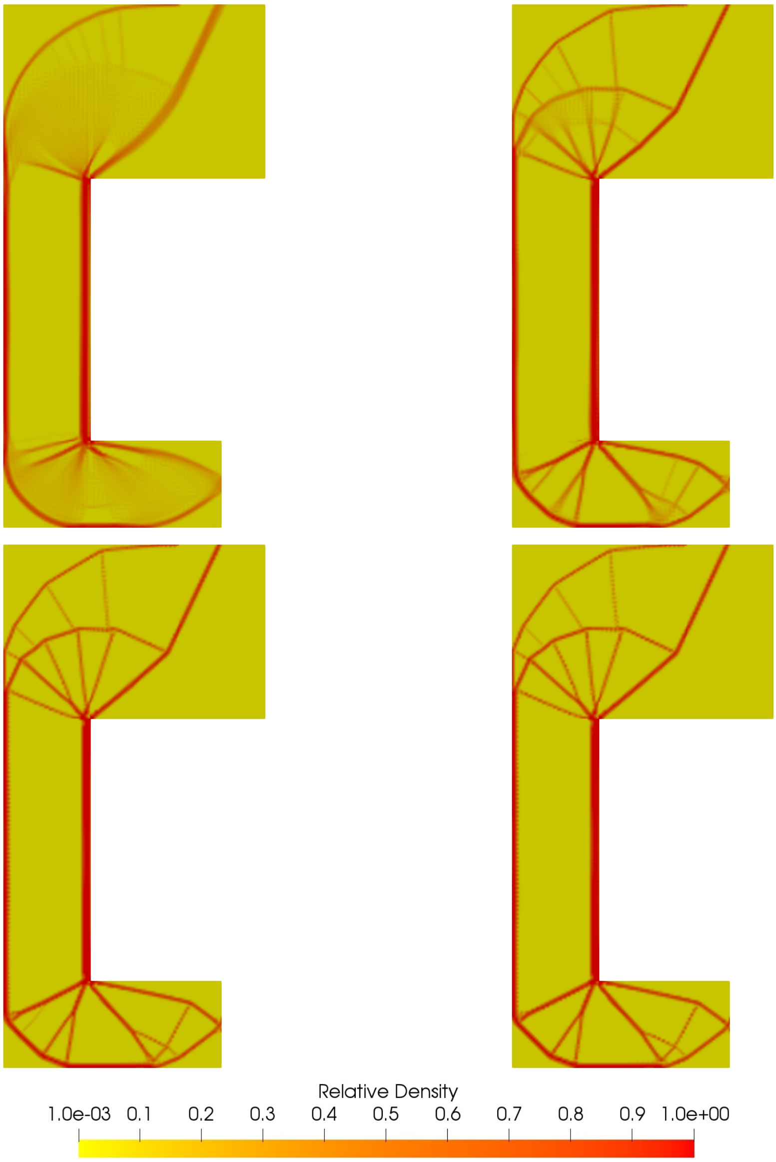


Figure 9.37. Undercarriage: Evolution of the relative density by using the Isogeometric Analysis [iterations 1400, 2100, 2800, 3500]

Tables 9.17, 9.18, 9.19 and 9.20 show the value of the most important parameters introduced in the topology optimization problem. The CPU time required to solve the problem with the Isogeometric Analysis is lower than with the conventional Finite Element Method.

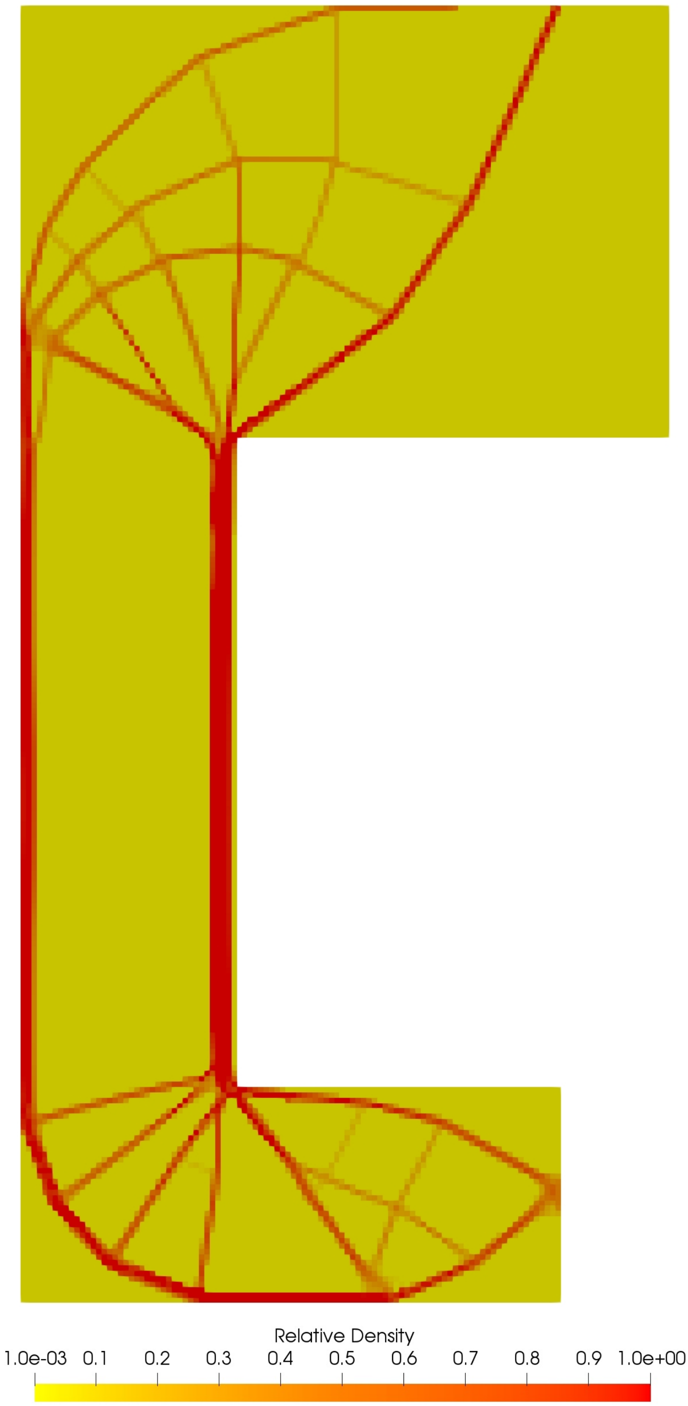


Figure 9.38. Undercarriage: Optimal solution by using the Finite Element Method

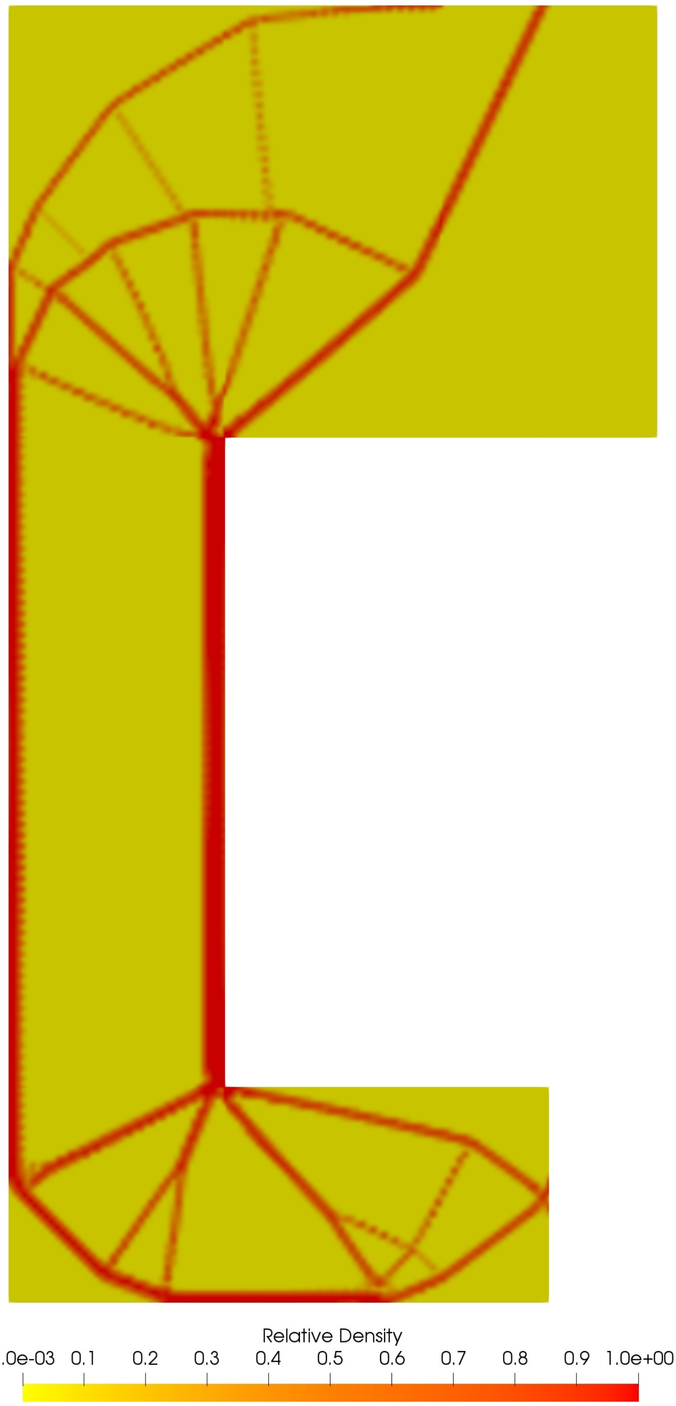


Figure 9.39. Undercarriage: Optimal solution by using the Isogeometric Analysis

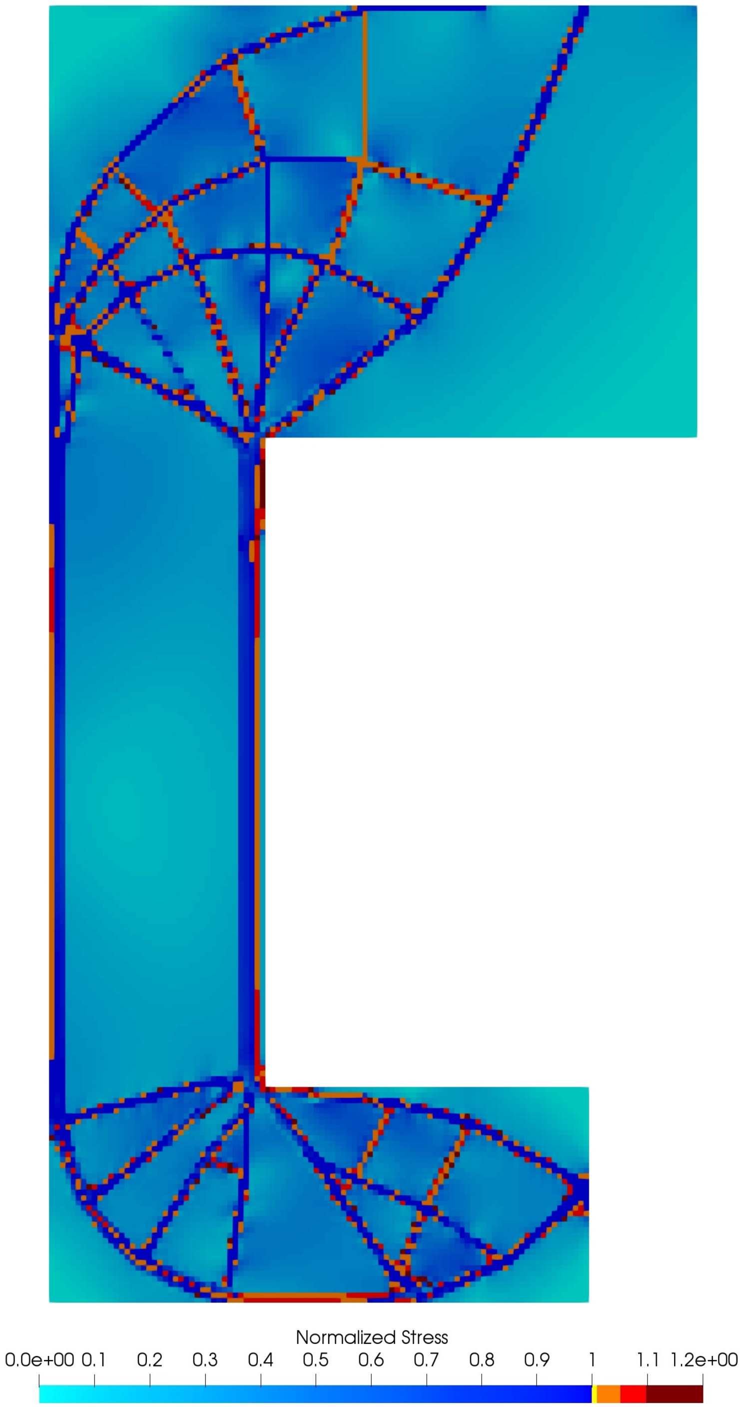


Figure 9.40. Undercarriage: Normalized stress by using the Finite Element Method

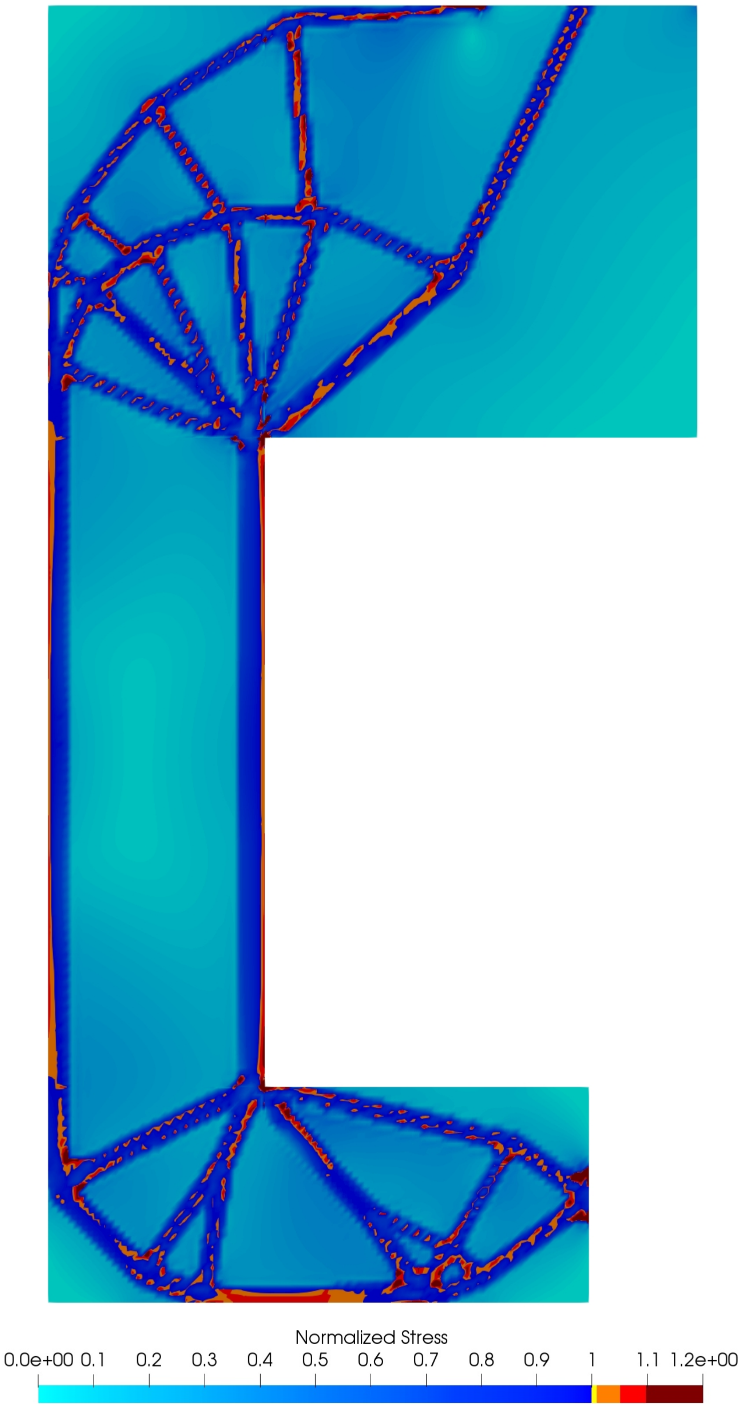


Figure 9.41. Undercarriage: Normalized stress by using the Isogeometric Analysis

	Finite Element Method	Isogeometric Analysis
Number of design variables	18400	19534
Number of constraints	1	1
Number of iterations	3500	3500
Final weight (kg)	1932.1	1985.5
CPU time (h)	18.69	11.07
Time per iteration (s)	19.3	11.4
Structural analysis time / Total time	74.58%	26.08%
Sensitivity analysis time / Total time	23.20%	65.11%

Table 9.17. Undercarriage: General parameters of the problem

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.18. Undercarriage: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-699	
2	0.001	700-1399	
3	0.002	1400-2099	
4	0.003	2100-2799	
5	0.005	2800-3500	

Table 9.19. Undercarriage: Value of the parameters during the resolution of the problem and range of application

The memory space required is basically the needed to store the structural stiffness matrix in both methods, since just one constraint is used to introduce the effect of local stress constraints. As a result of this, the Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis.

On the other hand, the number of iterations of the topology optimization algorithm used to obtain the final solution has been 3500 in both cases. However, it is possible to observe in figures 9.35 and 9.37 that there are no considerable differences between the solution obtained for a smaller number of iterations and the final solution. This circumstance allows to ensure that the solution has converged to its optimum.

Even though the ideal situation would be to stop the algorithm with the convergence of the solution, it is not suitable in this case due to the high non-linearity of the problem,

	Finite Element Method	Isogeometric Analysis
Initial moving limits of the design variables	0.005	
Factor of evolution of the moving limits between sets of iterations	0.75	
Number of iterations between two consecutive modifications	350	

Table 9.20. Undercarriage: Evolution of the moving of the design variables during optimization

especially in the Damage Constraint.

Furthermore, the final solutions obtained with both methods are equivalent in terms of topology, since their material layout is quite similar. The possible differences between both solutions are related to the material layout discretization. The final solution consists in a similar structure to the L-shaped beam obtained previously in the lower part of the domain and a certain set of bars that represent circumference arcs and several radius of these circumferences, being the point of contact among three regions placed in the upper part of the domain the center of the circumference, this part of the structure connects the L-shape beam and the supports.

Finally, it is important to remark that the amount of material required to manufacture the optimal solution obtained with both methods is similar since the difference is only the 3% of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout will not suppose an important difference in the results obtained with respect to its structural weight.

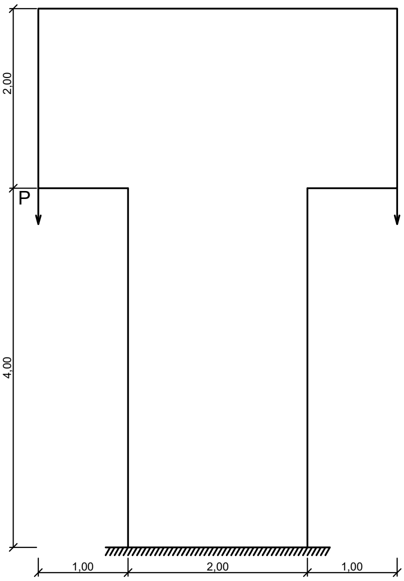
9.3.3. Mast

The third example of this section is a mast. This example has a rigid support in the lower border of the domain. That means null horizontal and vertical displacements on this border. Furthermore, two vertical loads will be applied in both lower corners of the lateral cantilevers.

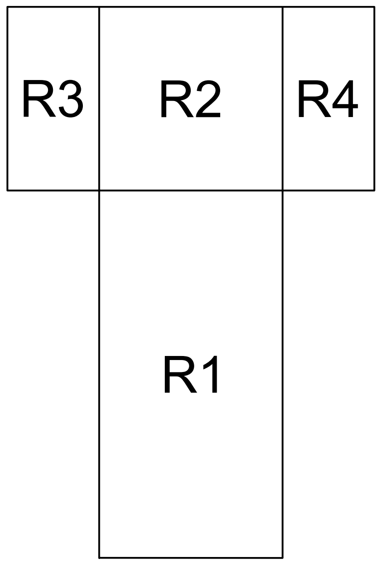
The dimensions of the domain used to solve this problem and the location of the external loads applied to the structure can be seen in figure 9.42a. Moreover, the structural weight will be considered as a structural load.

The structural domain will be discretized by means of 20736 quadratic serendipity elements with 8 nodes in the case of the conventional Finite Element Method and the same number of quadratic knot spans with 9 nodes in the case of the Isogeometric Analysis. On the other hand, the structural thickness is 0.175 m.

At this point, it is important to remark that the domain will have to be divided in 4 regions (figure 9.42b). This is due to the limitations of the Isogeometric Analysis when non-rectangular domains are required. In other words, only square or rectangular regions can be directly discretized with the Isogeometric Analysis. Additionally, it will be required compatibility between these regions, in terms of its dimensions, because of the continuity of the structure in the domain with respect to the relative density, the displacements field and the stress value in case of the Isogeometric Analysis.



(a) Problem statement. (units - m)



(b) Division of the domain in rectangular regions

Figure 9.42. Mast: Problem statement and domains discretization

Although the external loads applied over the structure are point loads, they will be distributed over 6 adjacent elements in order to avoid stress concentration phenomenon. The value of these point loads will be equal to 52 10^3 kN. The material used for the design of this structure is steel with: material density $\gamma_{mat} = 7850\ kg/m^3$, Young's modulus $E = 2.1\ 10^5\ MPa$, Poisson's ratio $\nu = 0.3$ and yield stress $\tilde{\sigma}_{max} = 230\ MPa$.

Figures 9.43, 9.44, 9.45 and 9.46 represent the evolution of the optimal design during the topology optimization process, and figures 9.47 and 9.48 show the final solution obtained with the formulations proposed in this thesis.

Figures 9.47 and 9.48 show that the solutions obtained with both methods coincides with the solutions obtained previously for the same problem in the literature. Apart from that, the main difference between both solutions is due to the material layout discretization of each method.

In addition, the most important characteristic of this example is the symmetry of the final solution between the left and the right half of the domain since the problem is completely symmetric. If, for instance, the value of both external loads did not coincide the final solution would not be symmetric.

On the other side, figures 9.49 and 9.50 represent the stress state by means of the normalized stress of the solutions proposed in figures 9.47 and 9.48. The value of the normalized stress is equal to the quotient between the stress in each point of the domain and the stress relaxation coefficient times the maximum allowable stress in that point.

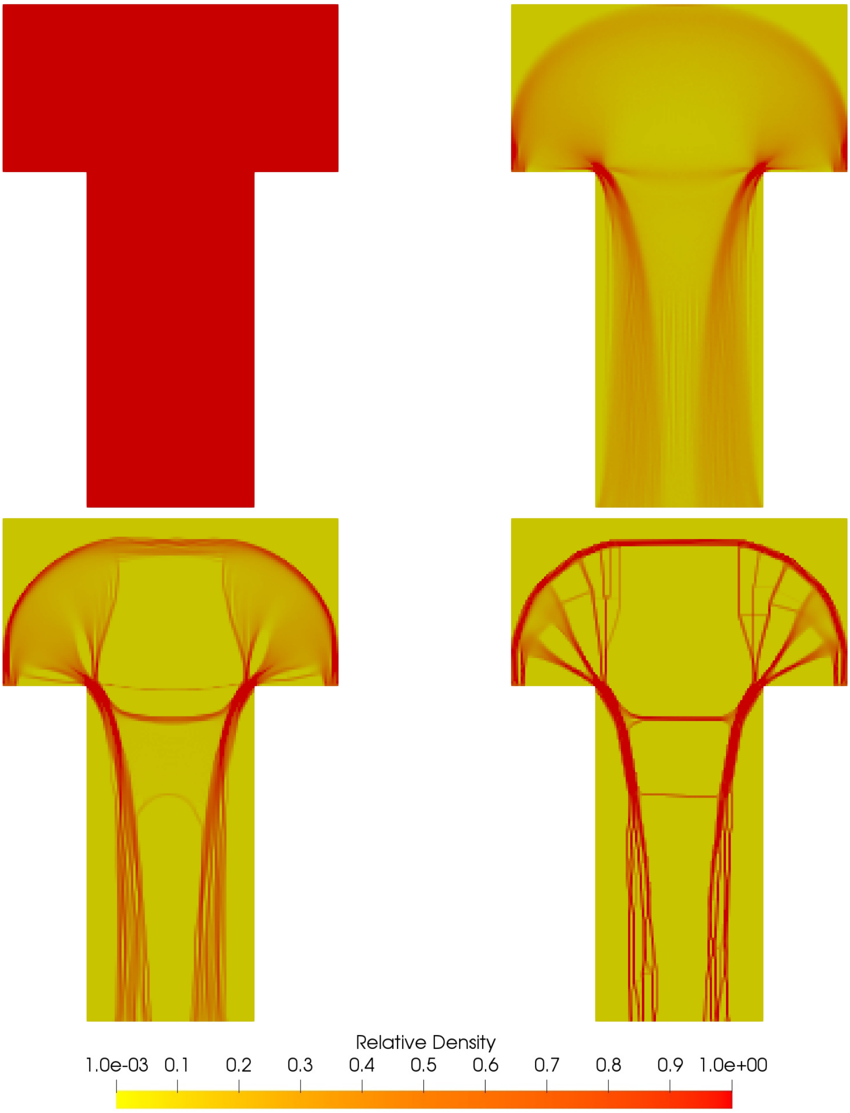


Figure 9.43. Mast: Evolution of the relative density by using the Finite Element Method [iterations 0, 1100, 2200, 3300]

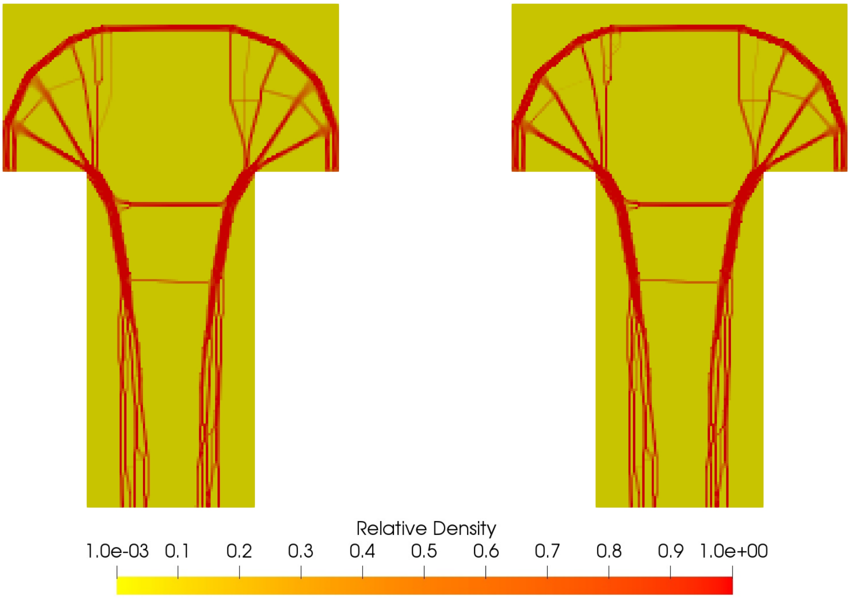


Figure 9.44. Mast: Evolution of the relative density by using the Finite Element Method [iterations 4400, 5500]

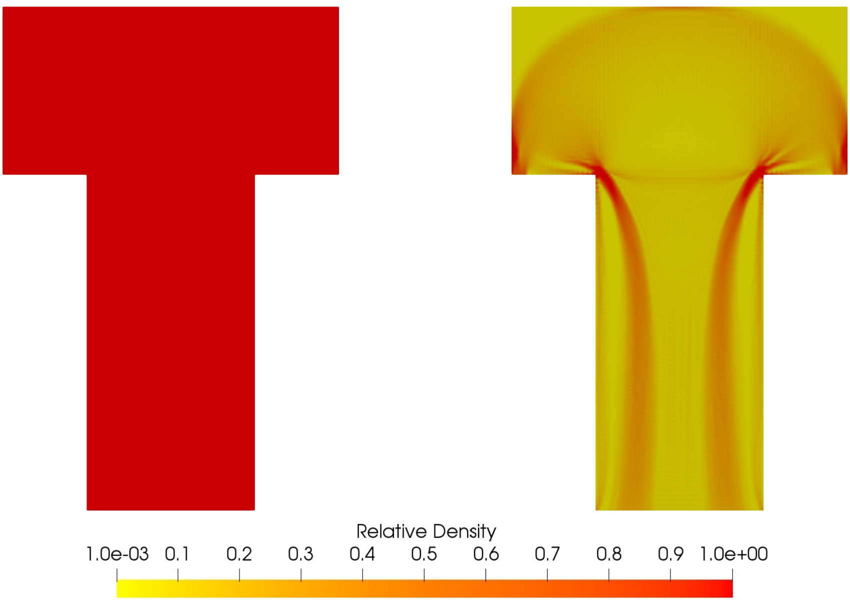


Figure 9.45. Mast: Evolution of the relative density by using the Isogeometric Analysis [iterations 0, 1600]

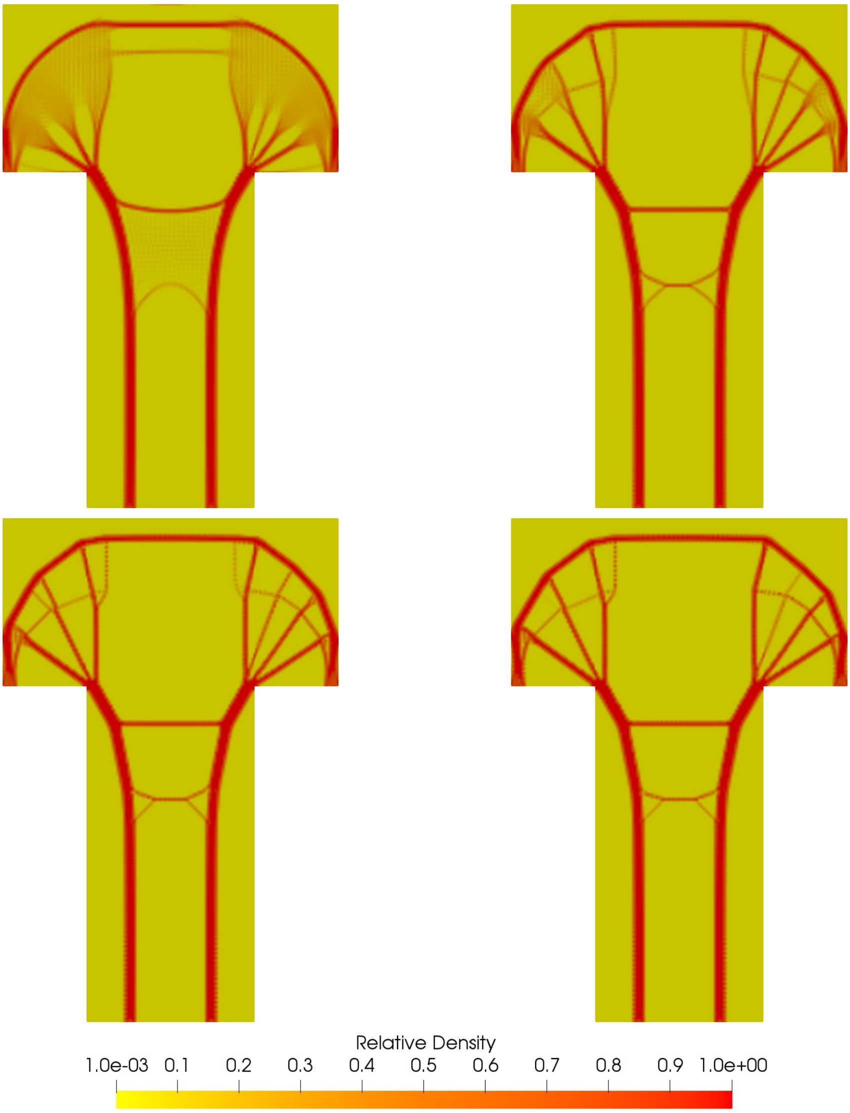


Figure 9.46. Mast: Evolution of the relative density by using the Isogeometric Analysis [iterations 3200, 4800, 6400, 8000]

The relaxation of the damage constraint produces the appearance of regions where the value of the stresses is slightly higher than their maximum allowable value that tend to coincide with the areas whose relative density is close to its lower limit, moreover, in this case and due to the use of the Multiregion Approach this phenomenon also happens in the proximities of the points of contact among three regions in that the domain had been divided because of the stress concentration, as it can be seen in figures 9.49 and 9.50.

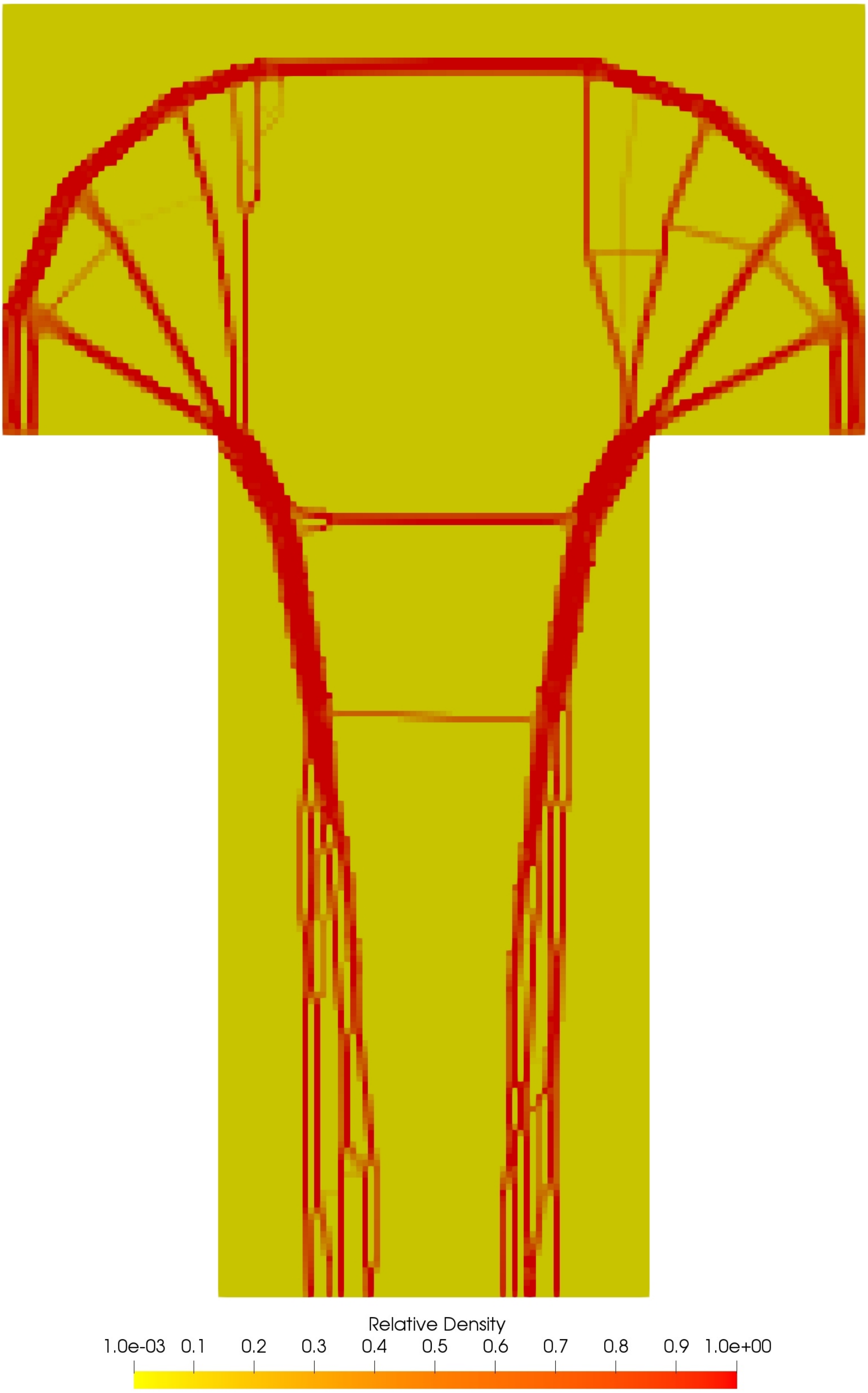


Figure 9.47. Mast: Optimal solution by using the Finite Element Method

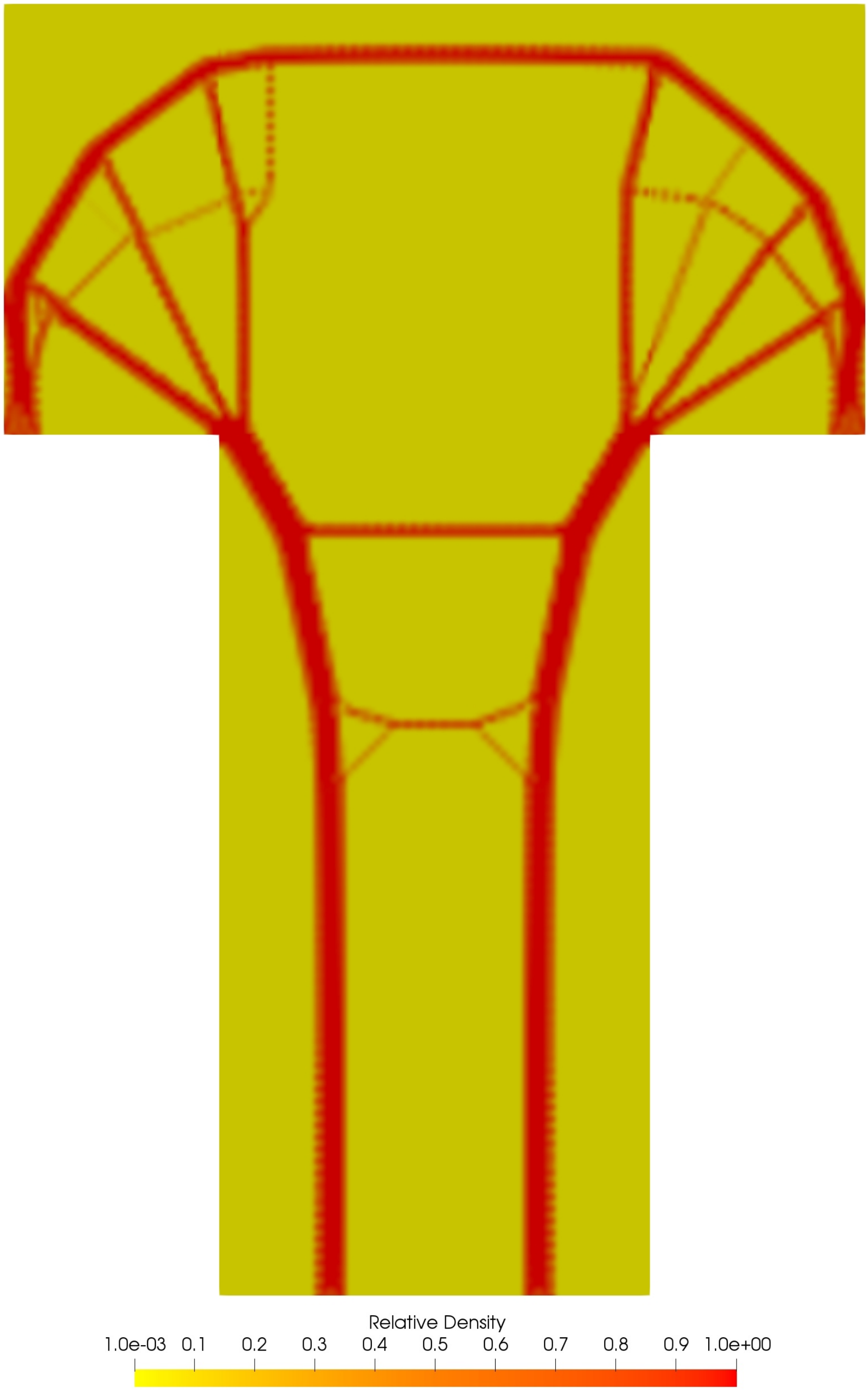


Figure 9.48. Mast: Optimal solution by using the Isogeometric Analysis

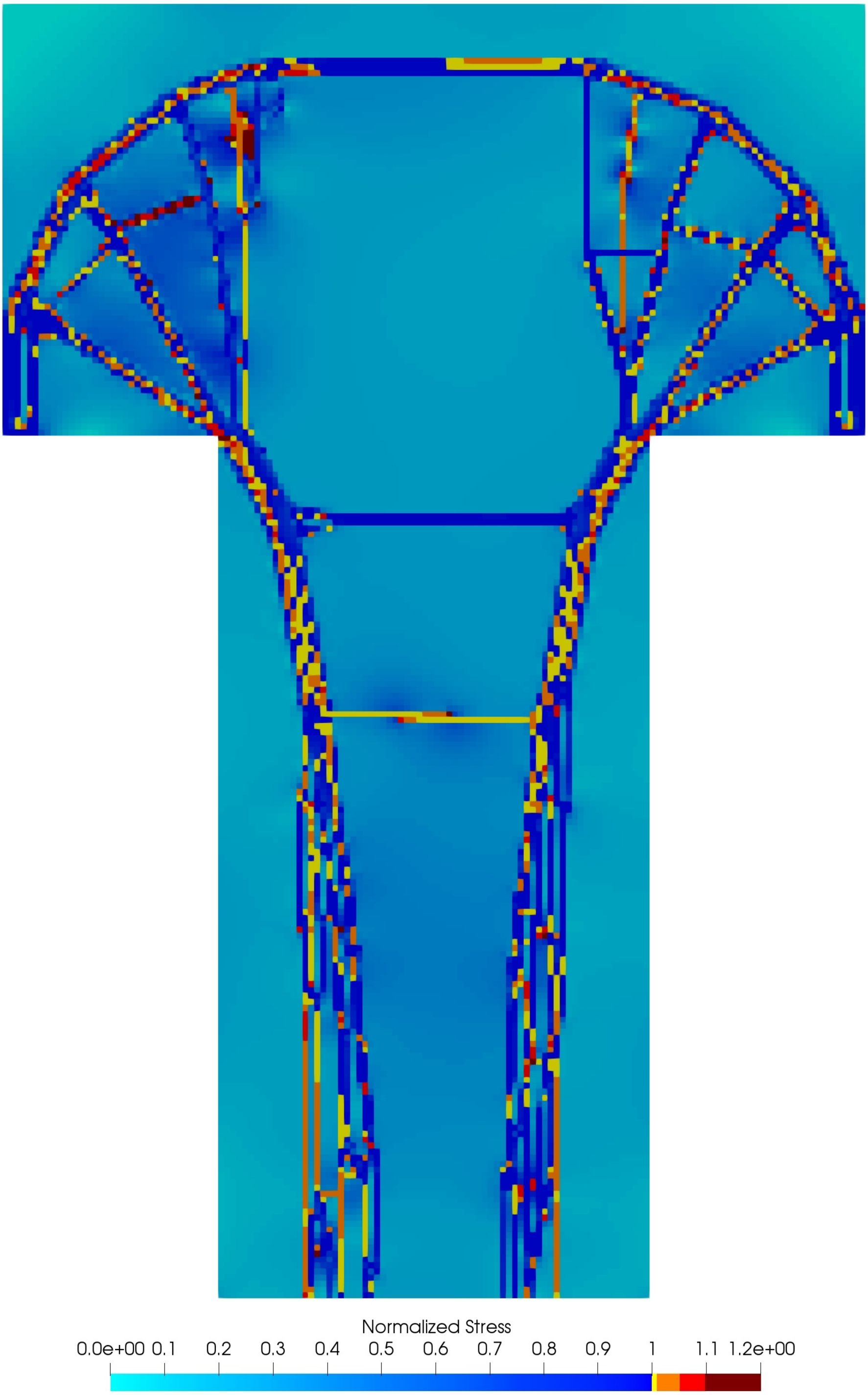


Figure 9.49. Mast: Normalized stress by using the Finite Element Method

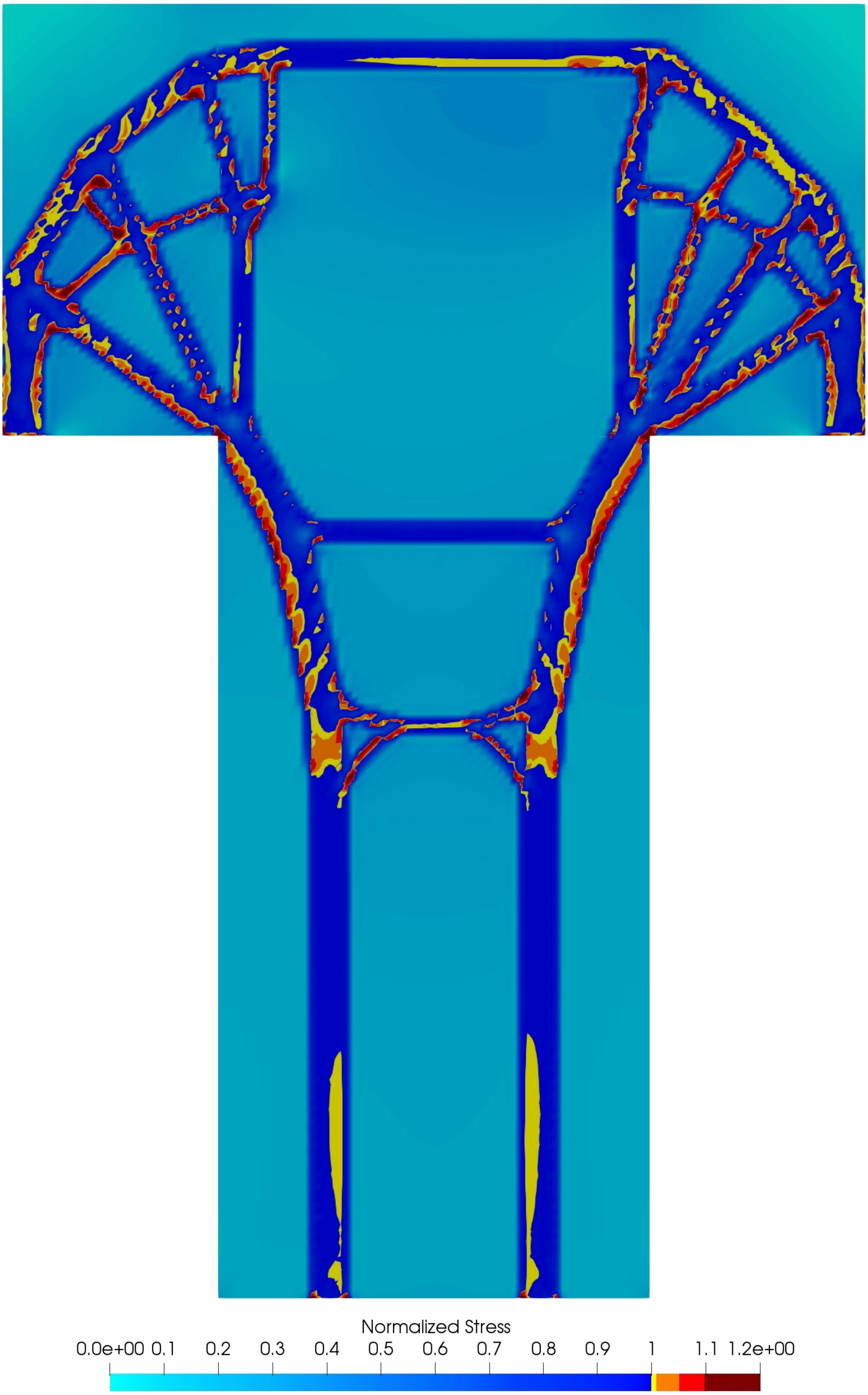


Figure 9.50. Mast: Normalized stress by using the Isogeometric Analysis

However, it is the definition of the damage constraint what produces the first case since the stresses can take whatever value if the relative density is equal to its lower limit and the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint.

	Finite Element Method	Isogeometric Analysis
Number of design variables	20736	21682
Number of constraints	1	1
Number of iterations	5500	8000
Final weight (kg)	3063.0	3041.4
CPU time (h)	51.26	37.00
Time per iteration (s)	33.6	16.7
Structural analysis time / Total time	81.63%	34.52%
Sensitivity analysis time / Total time	16.77%	57.79%

Table 9.21. Mast: General parameters of the problem

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.22. Mast: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-1099	0-1599
2	0.001	1100-2199	1600-3199
3	0.002	2200-3299	3200-4799
4	0.003	3300-4399	4800-6399
5	0.005	4400-5500	6400-8000

Table 9.23. Mast: Value of the parameters during the resolution of the problem and range of application

Tables 9.21, 9.22, 9.23 and 9.24 show the most important parameters of the problem. It is possible to observe that the CPU time required to solve the problem with the Isogeometric Analysis is lower than with the conventional Finite Element Method despite of having to use more iterations to obtain the solution with the Isogeometric Analysis.

	Finite Element Method	Isogeometric Analysis
Initial moving limits of the design variables	0.005	
Factor of evolution of the moving limits between sets of iterations	0.75	
Number of iterations between two consecutive modifications	550	800

Table 9.24. Mast: Evolution of the moving of the design variables during optimization

The memory space required is basically the needed to store the structural stiffness matrix in both methods, since just one constraint is used to introduce the effect of local stress constraints. As a result of this, the Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis. This circumstance has been commented in Chapter 8.

On the other hand, the number of iterations used has been 5500 in case of the conventional Finite Element Method and 8000 in case of the Isogeometric Analysis. However, it is possible to observe in figures 9.44 and 9.46 that there are few differences between the solution obtained for a smaller number of iterations and the final solution. This circumstance allows to ensure that the solution has converged to its optimum.

The use of a linear approximation for the Damage Constraint in the solution of the problem means the non-existence of convergence properly since the Damage Constraint is extremely non-linear. For this reason, the algorithm will not stop because of the convergence of the solution what would be the ideal situation unless a comparison between two different solutions will be established as a way to analyze the convergence.

Furthermore, the final solution obtained with both methods is equivalent in terms of topology, since the global material layout is equivalent. The possible differences between both solutions are related to the material layout discretization. The final solution consists in two vertical bars that connect the supports with the upper part of the domain, a set of horizontal bars that connect the left and the right half of the structure and a set of bars that represent both circumference arches and radius of these circumferences, being the points of contact among three regions the center of these circumferences, they are placed in the cantilevers. The vertical bars can be seen properly in the solution obtained with the Isogeometric Analysis because of impossibility of developing sharp transitions from low densities to high densities, the Finite Element Method is not able to develop it properly, since the structures submitted to traction or compression pure forces does not have only one solution.

Finally, it is important to remark that the amount of material required to manufacture the optimal solution obtained with both methods is similar since the difference is only the 0.1% of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout will not suppose an important difference in the results obtained with respect to its structural weight.

9.3.4. MBB beam

The last example of this section is a MBB beam. This is a structure used commonly in the aeronautic field to resist all the deck loads in a plane and to transfer them to the fuselage. This beam receives its name from the German company that designed it initially: Messerschmitt-Bölkow-Blohm. This beam is mounted on 2 supports, with horizontal displacements allowed but with vertical displacements suppressed. This example has its geometric characteristics perfectly defined and a vertical distributed load will be applied in the central part of the structure over the upper edge.

At this point, it is important to remark that it will be possible to calculate only one half of the structure since the structural problem is symmetric. In this case, the right half of the structure will be analyzed. As a result of this it will be necessary to modify the structural supports. It will be necessary to introduce null horizontal displacements in the symmetry axis.

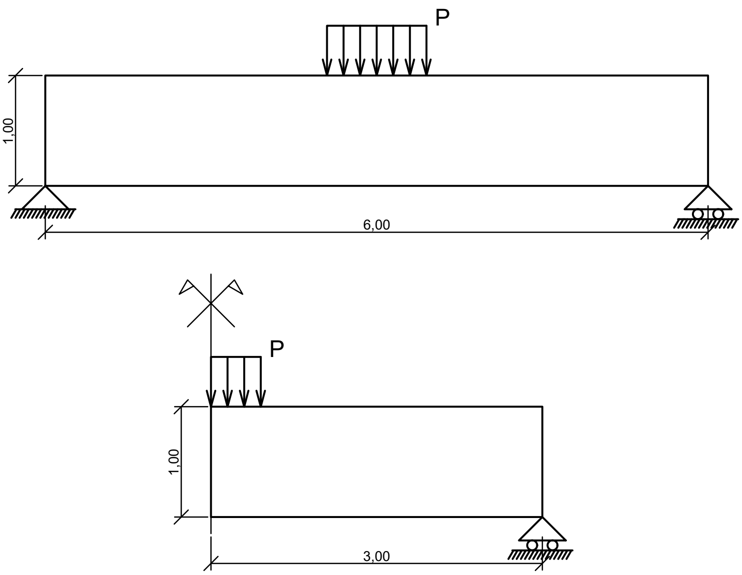


Figure 9.51. MBB beam: Problem statement. (units - m)

Figure 9.51 shows a graphical scheme with the dimensions of the entire structure, the part of the structure that will be object of optimization and the position of the external loads applied. Moreover, it is important to establish that the structural weight will be considered as a structural load.

The structural domain considered will be discretized by means of a regular mesh of 240 x 80 = 19200 quadratic serendipity elements with 8 nodes in the case of the Finite Element Method and the same number of quadratic knot spans with 9 nodes in the case of the Isogeometric Analysis. On the other hand, the structural thickness will be in this case 1.25 m.

An external distributed load of 160 10^3 kN/m is applied over 32 adjacent elements in the superior part of the domain in the proximity of the symmetry axis. The material used for the design of this structure will be steel with the same properties of previous examples.

Figures 9.52, 9.53, 9.54 and 9.55 represent the evolution of the optimal design during the topology optimization procedure, and figures 9.56 and 9.57 show the final solution obtained.

As it can be seen in figures 9.56 and 9.57, the solution obtained with both methods coincides with the solutions obtained previously in the literature for the same problem. Apart from that, the main difference between both solutions is due to the material layout definition with each method.

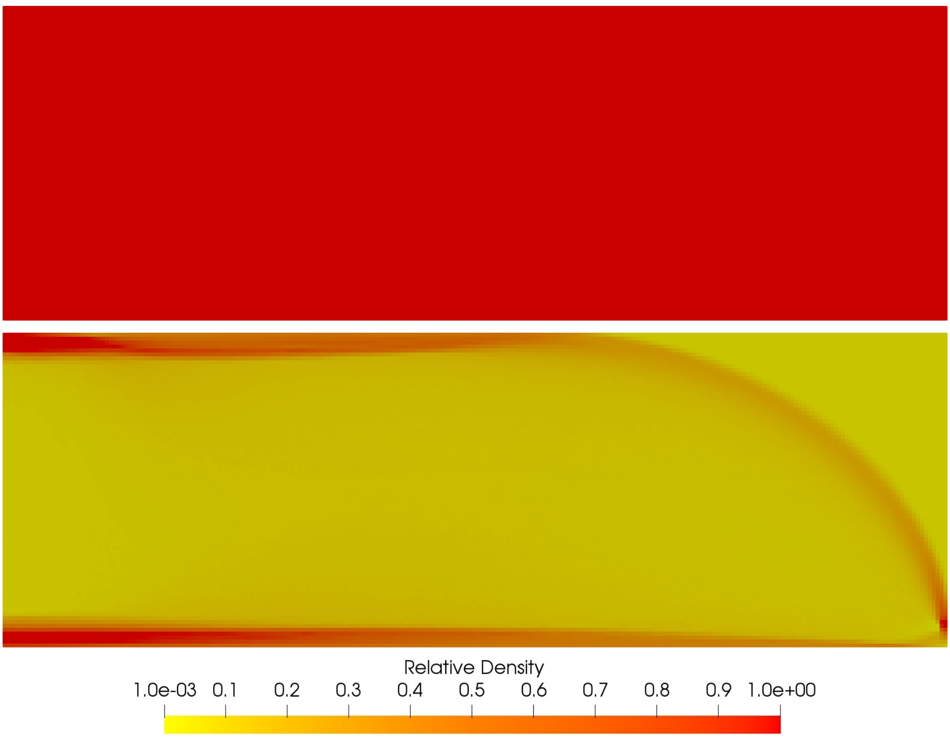


Figure 9.52. MBB beam: Evolution of the relative density by using the Finite Element Method [iterations 0, 700]

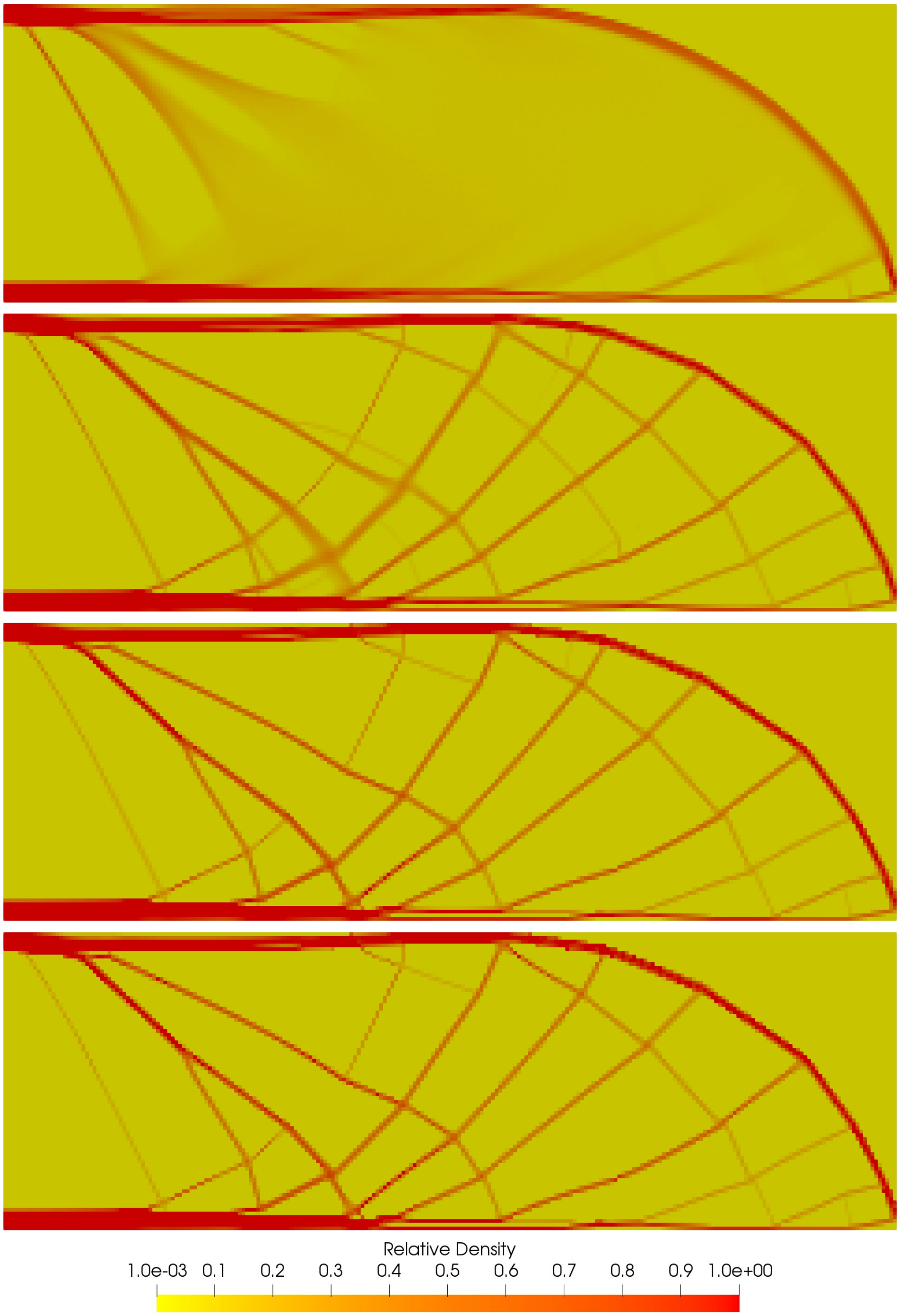


Figure 9.53. MBB beam: Evolution of the relative density by using the Finite Element Method [iterations 1400, 2100, 2800, 3500]

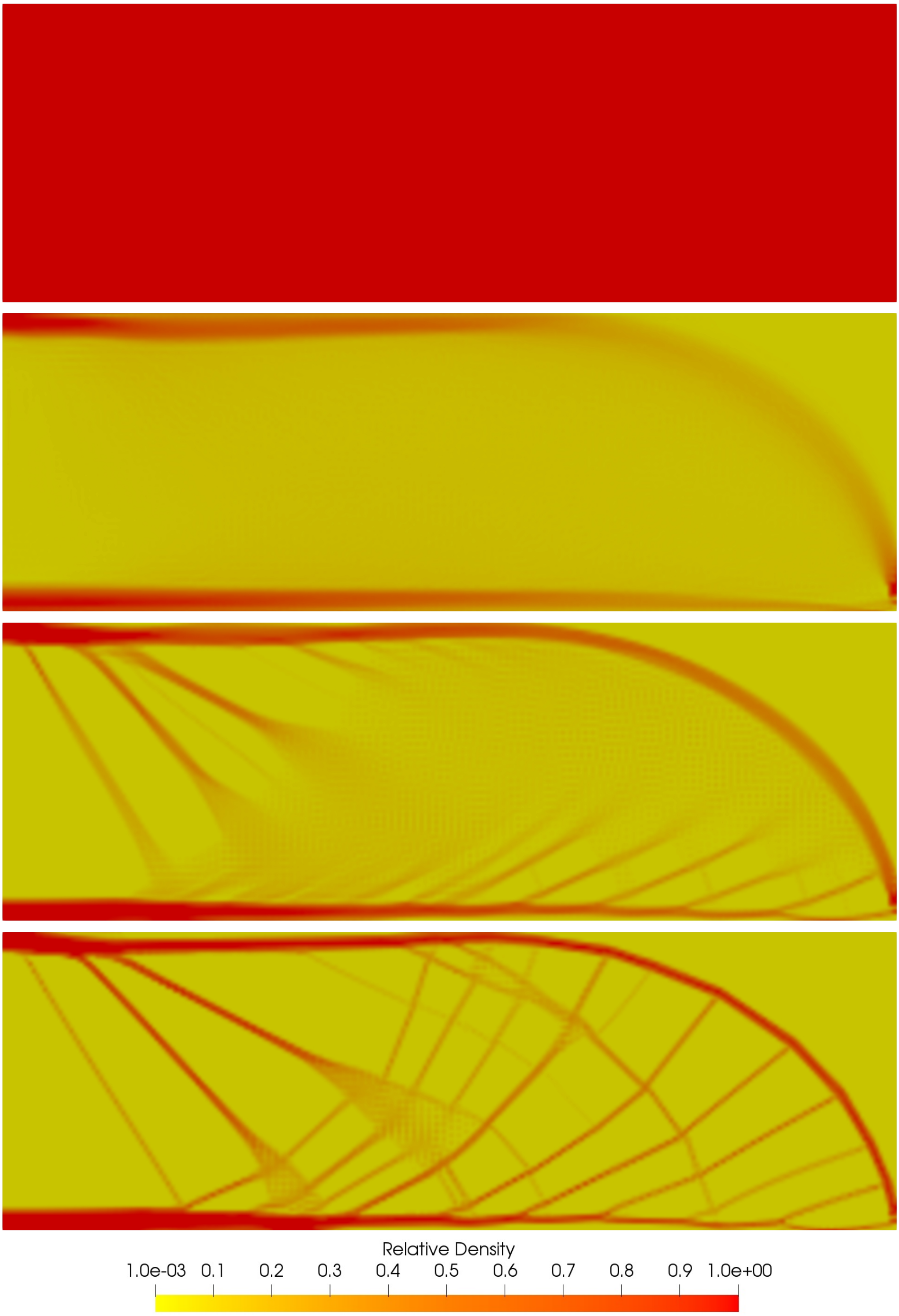


Figure 9.54. MBB beam: Evolution of the relative density by using the Isogeometric Analysis [iterations 0, 700, 1400, 2100]

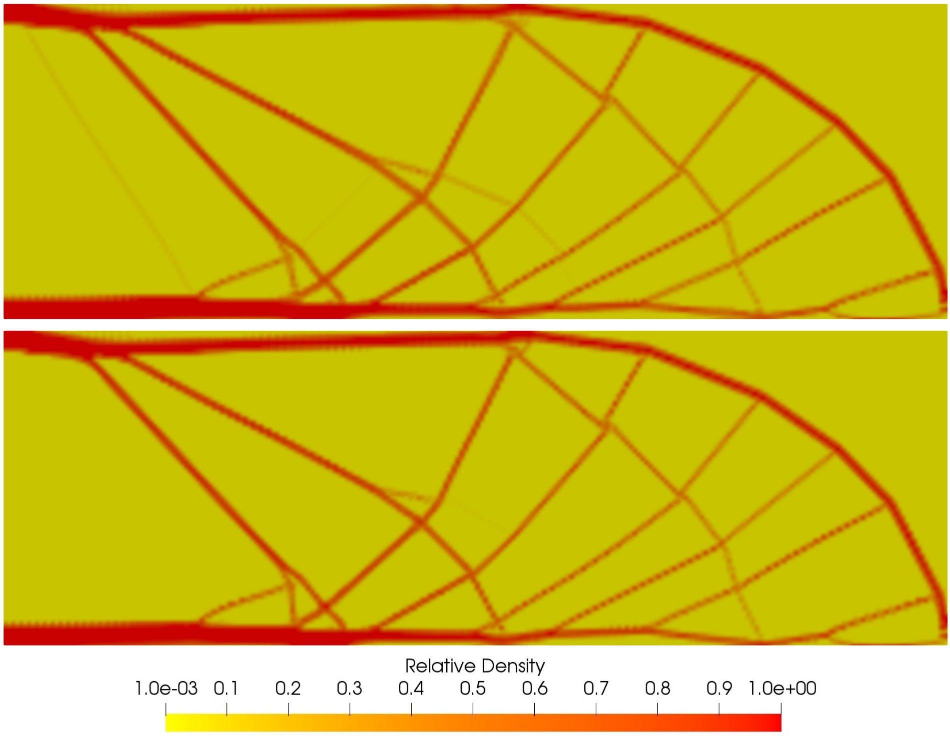


Figure 9.55. MBB beam: Evolution of the relative density by using the Isogeometric Analysis [iterations 2800, 3500]

On the other hand, figures 9.58 and 9.59 represent the stress state by means of the normalized stress of the solutions obtained with the different formulations proposed in this thesis. The normalized stress is obtained through the quotient between the stress in each point of the domain and the stress relaxation coefficient times the maximum allowable stress in that point.

Figures 9.58 and 9.59 show that there are regions whose stress value is slightly higher than their maximum allowable value. Nevertheless, this circumstance is due to the relaxation of the damage constraint, and tends to appear in areas with relative density close to its lower limit since the own definition of the damage constraint implies that the stresses can take whatever value if the relative density is equal to its lower limit and the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint.

At this point, it is important to remark that the real solution of the problem proposed can be obtained by means of the symmetrical replication of the solution shown in 9.56 and 9.57 through the symmetry axis.

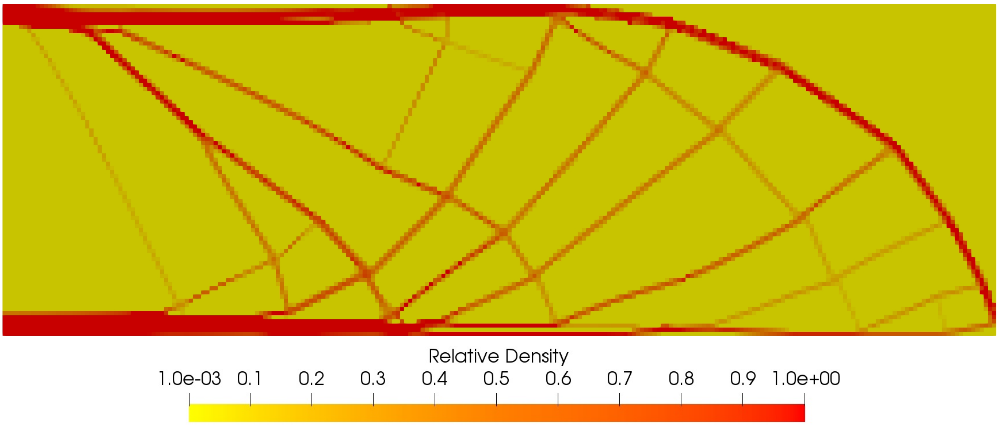


Figure 9.56. MBB beam: Optimal solution by using the Finite Element Method

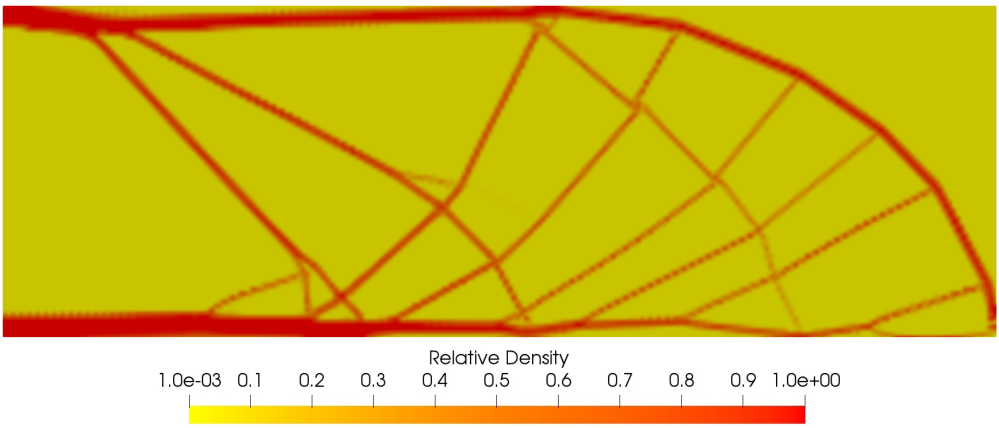


Figure 9.57. MBB beam: Optimal solution by using the Isogeometric Analysis

Tables 9.25, 9.26, 9.27 and 9.28 show the value of the most important parameters introduced in the topology optimization problem. The CPU time required to solve the problem with the Isogeometric Analysis is lower than with the conventional Finite Element Method.

The memory space required is small, basically just the needed to store the structural stiffness matrix in both methods due to the use of only one constraint to introduce the effect of local stress constraints. As a result of this, the conventional Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis formulation. This circumstance is due to the lower number of points required in the solution of the structural analysis what is related with the size of the global stiffness matrix and a lower bandwidth of the global stiffness matrix in case of the Isogeometric Analysis.

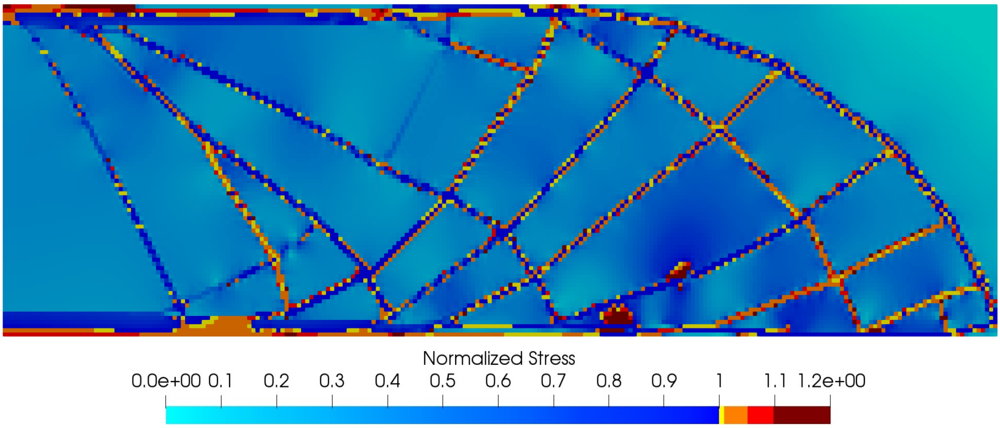


Figure 9.58. MBB beam: Normalized stress by using the Finite Element Method

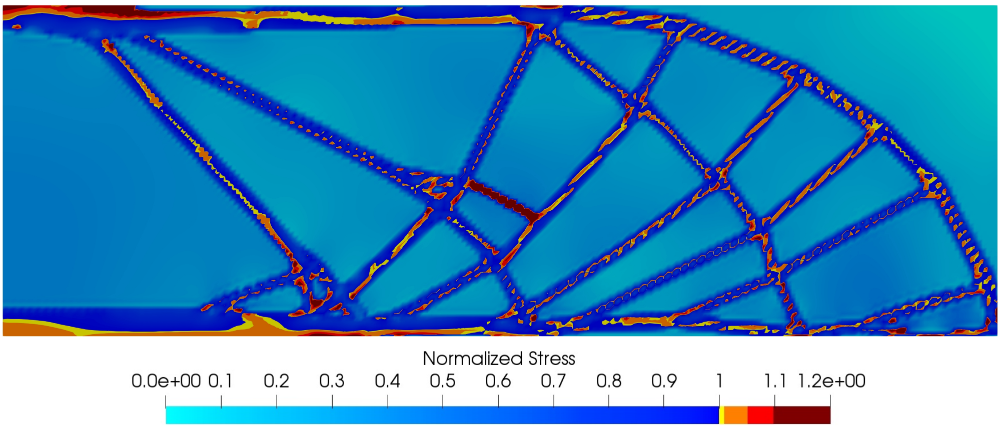


Figure 9.59. MBB beam: Normalized stress by using the Isogeometric Analysis

Moreover, the number of iterations of the topology optimization algorithm used to obtain the final solution has been 3500 in both cases. Nonetheless, it is possible to observe in figures 9.53 and 9.55 that there are few differences between the solution obtained for a smaller number of iterations and the final solution. The use of more iterations that the needed allows to ensure the convergence of the solution to its optimum.

Even though the ideal situation would be to stop the algorithm due to the convergence of the solution, this convergence does not exist properly due to the high non-linearity of the problem, especially in case of the Damage Constraint, since the approximation used in the solution of the problem for the Damage Constraint is linear.

	Finite Element Method	Isogeometric Analysis
Number of design variables	19200	19844
Number of constraints	1	1
Number of iterations	3500	3500
Final weight (kg)	3146.0	3265.8
CPU time (h)	15.92	11.98
Time per iteration (s)	16.4	12.4
Structural analysis time / Total time	66.81%	23.33%
Sensitivity analysis time / Total time	30.13%	66.95%

Table 9.25. MBB beam: General parameters of the problem

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.26. MBB beam: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-699	
2	0.001	700-1399	
3	0.002	1400-2099	
4	0.003	2100-2799	
5	0.005	2800-3500	

Table 9.27. MBB beam: Value of the parameters during the resolution of the problem and range of application

On the other side, the final solutions obtained with both methods are equivalent in terms of topology, since the global material layout is similar. The final solution consists in a certain set of bars. Two of these bars are horizontal and they are placed in the upper and the lower part of the domain. The rest of the bars connect this two bars and they tend to coincide with the isostatic lines.

Nevertheless, it is possible that the number of bars and their positions will be different depending on the method employed since the material layout discretization is influenced by it. In the conventional Finite Element Method, the value of the relative density at each point of an element is directly related with only one design variable, nonetheless, in the Isogeometric Analysis this relationship is more complex, since this value is a combination of design variables.

The use of a combination of design variables to define the material layout dis-

	Finite Element Method	Isogeometric Analysis
Initial moving limits of the design variables	0.005	
Factor of evolution of the moving limits between sets of iterations	0.75	
Number of iterations between two consecutive modifications	350	

Table 9.28. MBB beam: Evolution of the moving of the design variables during optimization

cretization with the Isogeometric Analysis means that when the penalty coefficient is introduced in the formulation of the problem the design variables whose value is close to their minimum allowable value tend to pull the nearest design variables to their minimum allowable value unless that the value of these design variables is not close enough to their minimum allowable value. This circumstance tends to happen when the value of most design variables that surround one of them is equal to their minimum allowable value. Consequently, the softer bars of the conventional Finite Element Method, what means a value of the relative density close to its lower limit, tend to disappear in the Isogeometric Analysis.

Finally, it is important to remark that the difference regarding the amount of material required to manufacture the optimal solution obtained with both methods is quite small since the difference is only the 4% of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout will not mean an important difference in the results obtained with respect to its structural weight.

9.4. Three-dimensional examples

In the previous sections, a number of different topology optimization problems in the two-dimensional space have been solved. Thus, the functioning of the topology optimization algorithm developed in this thesis has become validated with the exception of the three-dimensional structural analysis that has not been possible to test with the two-dimensional examples.

Consequently, two problems in the three-dimensional space will be solved in this section: a plate with a large height and four upper loads and a cantilever beam. These examples have been chosen since they are an extent of the two first examples solved in the two-dimensional space to the three-dimensional one, but with different conditions of load and support along the new dimension.

9.4.1. Plate with large height and four upper loads

The first example of this section is a plate with large height. This example will have fixed supports in the four lower corners what will mean null displacements in the three spatial directions. The main reason to include this example in this section is due to its solution is known, since there is an analogy between this example and the beam with large height and upper load solved previously. Although both examples are analogous, it is not possible to solve the problem proposed in this section in the two-dimensional space because the loads applied and the supports are not contained in the same plane. For this reason, it will be possible to test the formulation proposed in this thesis to solve the topology optimization problem of structures in the three-dimensional space.

First, figure 9.60 shows a graphical scheme with the dimensions of the domain of this problem and the position of the external loads applied. Moreover, it is important to establish that the structural weight will be considered as a structural load.

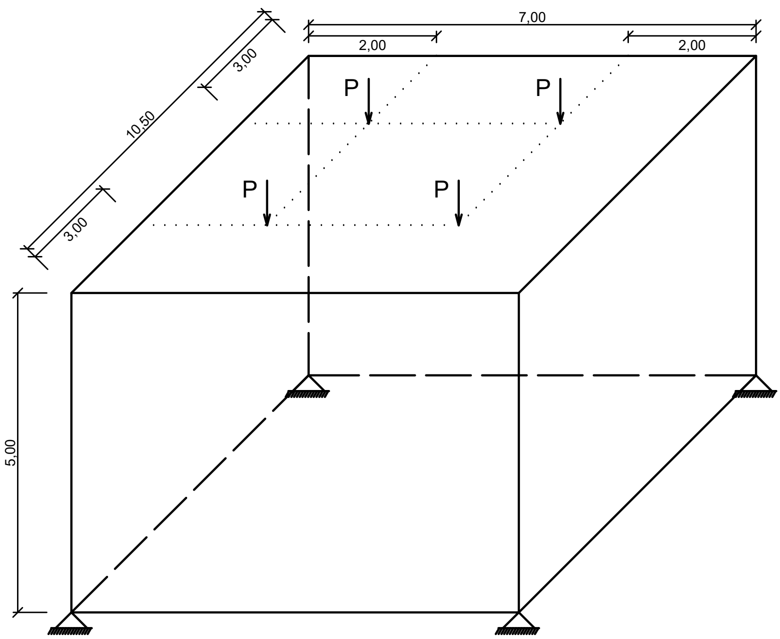


Figure 9.60. Plate with large height and four upper loads: Problem statement. (units - m)

The structural domain will be discretized by means of a regular mesh of 14 x 10 x 21 = 2940 quadratic serendipity elements with 20 nodes in the case of the conventional Finite Element Method and the same number of quadratic knot spans with 27 nodes in the case of the Isogeometric Analysis.

On the other hand, the external loads are four punctual forces. In order to avoid the stress concentration phenomenon, the forces will be applied in the contact point among four adjacent elements. The value of each punctual force will be 1240 10^3 kN. In the same way that for the two-dimensional examples, the material used for the design of this structure is steel with: material density $\gamma_{mat} = 7850\ kg/m^3$, Young's modulus $E = 2.1\ 10^5\ MPa$, Poisson's ratio $\nu = 0.3$ and yield stress $\tilde{\sigma}_{max} = 230\ MPa$.

Figures 9.61, 9.62, 9.63, 9.64, 9.65 and 9.66 show the evolution of the optimal design. Only the parts of the domain whose relative density is greater than or equal to 0.1 are plotted. Figures 9.67 and 9.69 represent the part of the optimal solution whose relative density is greater than or equal to 0.1. Finally, and with the objective to improve the representation of the optimal solution, figures 9.67, 9.68, 9.69 and 9.70 show the part of the domain whose relative density is greater than or equal to 0.3, 0.5 and 0.7 respectively.

According to figures 9.67, 9.68, 9.69 and 9.70, the solution obtained with both methods coincide with the theoretical solution. On the other side, the main difference between both solutions is due to the material layout definition with each method.

Figures 9.71, 9.72, 9.73 and 9.74 represent the stress state. For this purpose, it will be only plotted the regions of the optimal solutions whose normalized stress is greater than or equal to a certain limit. The normalized stress is obtained through the quotient between the stress in each point of the domain and the stress relaxation coefficient times the maximum allowable stress in that point.

Figures 9.72 and 9.74 show that there are regions whose stress value is slightly higher than their maximum allowable value. Nevertheless, this circumstance is due to the relaxation of the damage constraint, and tends to appear in the areas with relative density close to its lower limit since the own definition of the damage constraint implies that the stresses can take whatever value if the relative density is equal to its lower limit and the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint.

In addition, the most important characteristic of this example is the existence of symmetry in the final solution obtained between the left and the right part of the domain and between the front and the back part of the domain. This is the expected solution since the geometry and the load configuration is also symmetric. If a non-symmetric configuration is applied the final solution would not be symmetric.

Tables 9.29, 9.30, 9.31 and 9.32 indicate the value of the most important parameters introduced in the topology optimization problem. It can be observed regarding the computational requirements that in contrast to the two-dimensional examples the CPU time required to solve this three-dimensional problem with the Isogeometric Analysis proposed is higher than with the conventional Finite Element Method. However, this is due to the larger number of iterations required in the solution of the problem with the Isogeometric Analysis proposed, since its time per iteration is lower than with the conventional Finite Element Method.

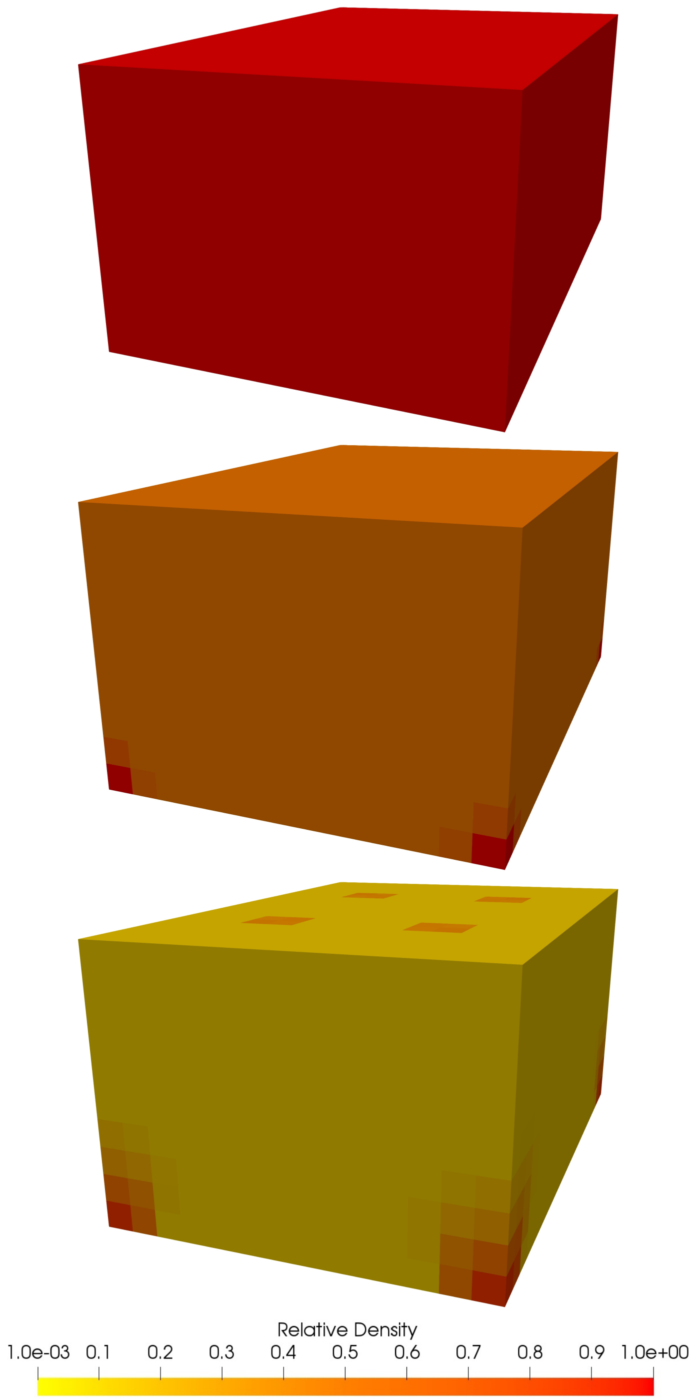


Figure 9.61. Plate with large height and four upper loads: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Finite Element Method [iterations 0, 100, 200]

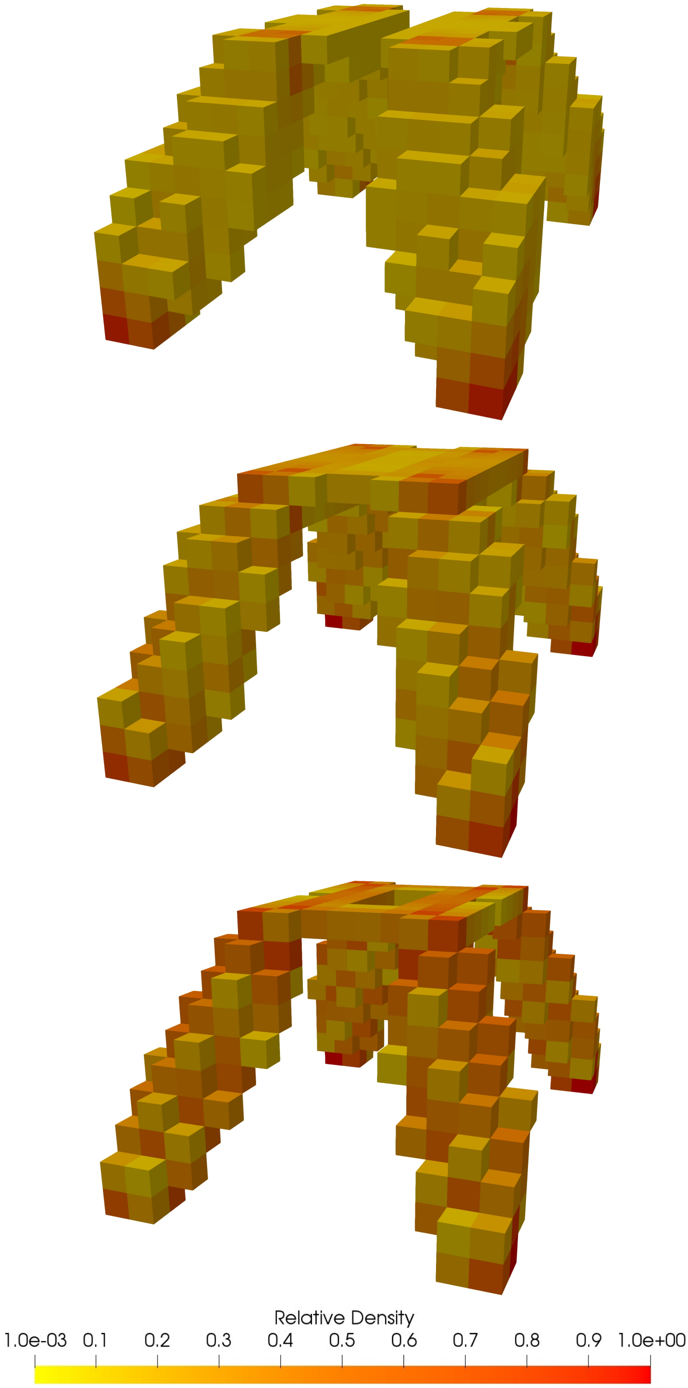


Figure 9.62. Plate with large height and four upper loads: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Finite Element Method [iterations 250, 300, 400]

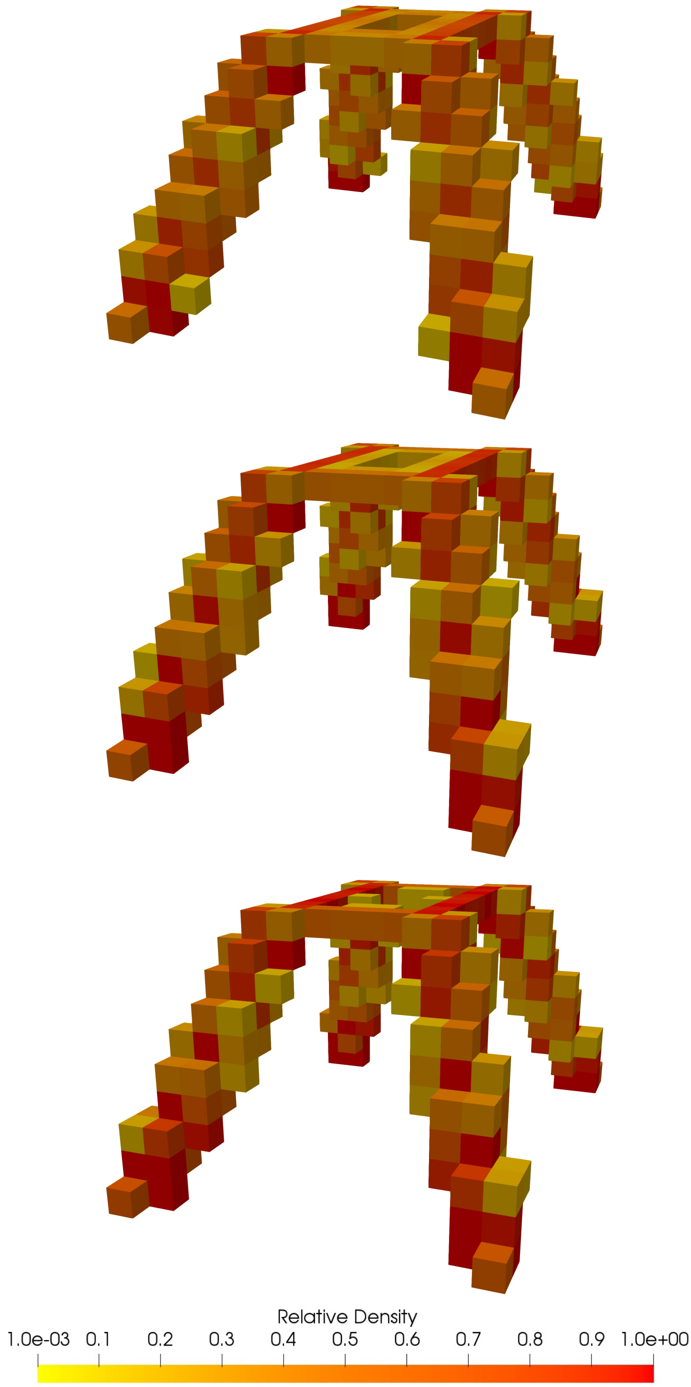


Figure 9.63. Plate with large height and four upper loads: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Finite Element Method [iterations 600, 800, 1000]

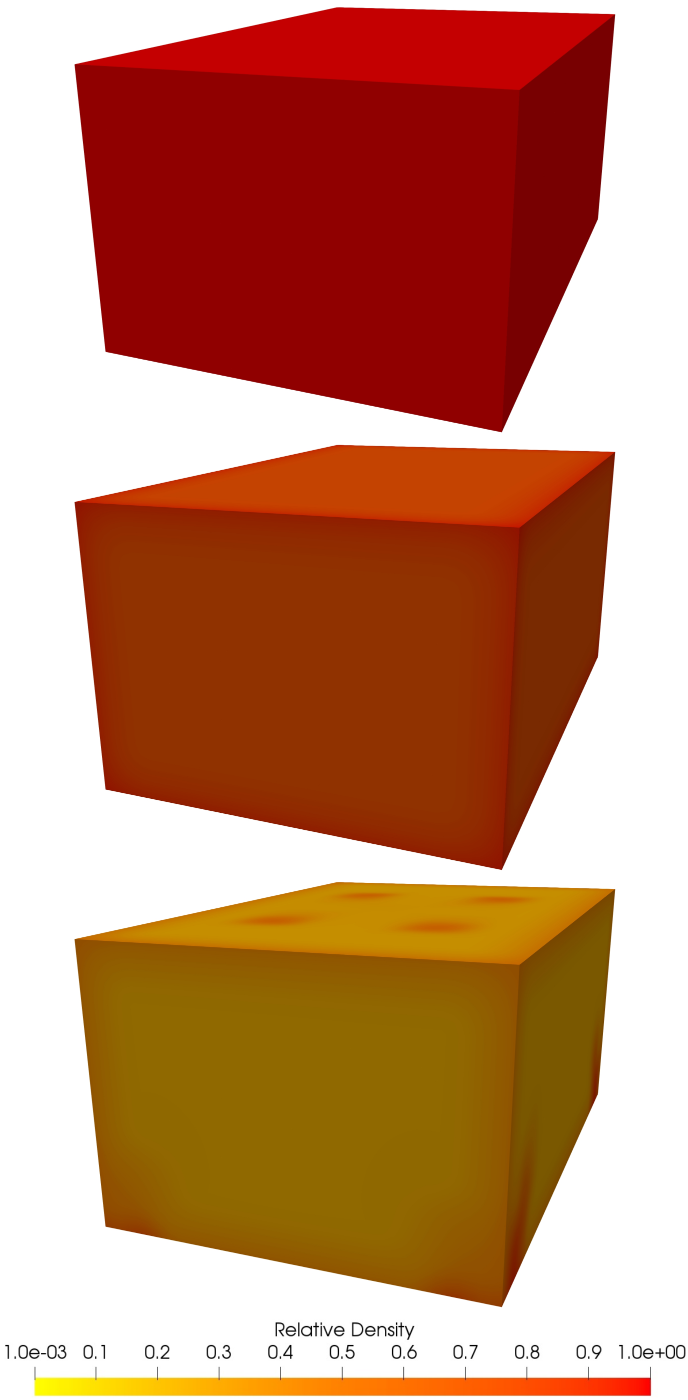


Figure 9.64. Plate with large height and four upper loads: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Isogeometric Analysis [iterations 0, 150, 300]

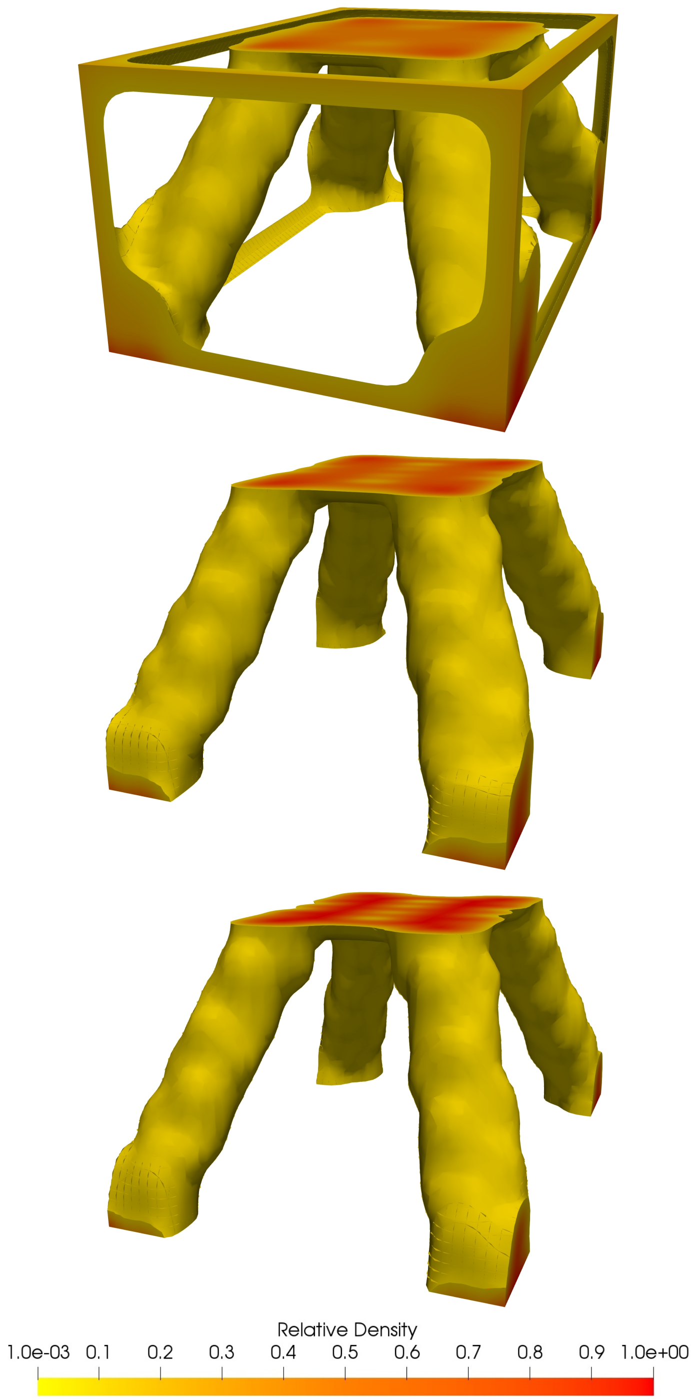


Figure 9.65. Plate with large height and four upper loads: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Isogeometric Analysis [iterations 375, 450, 600]

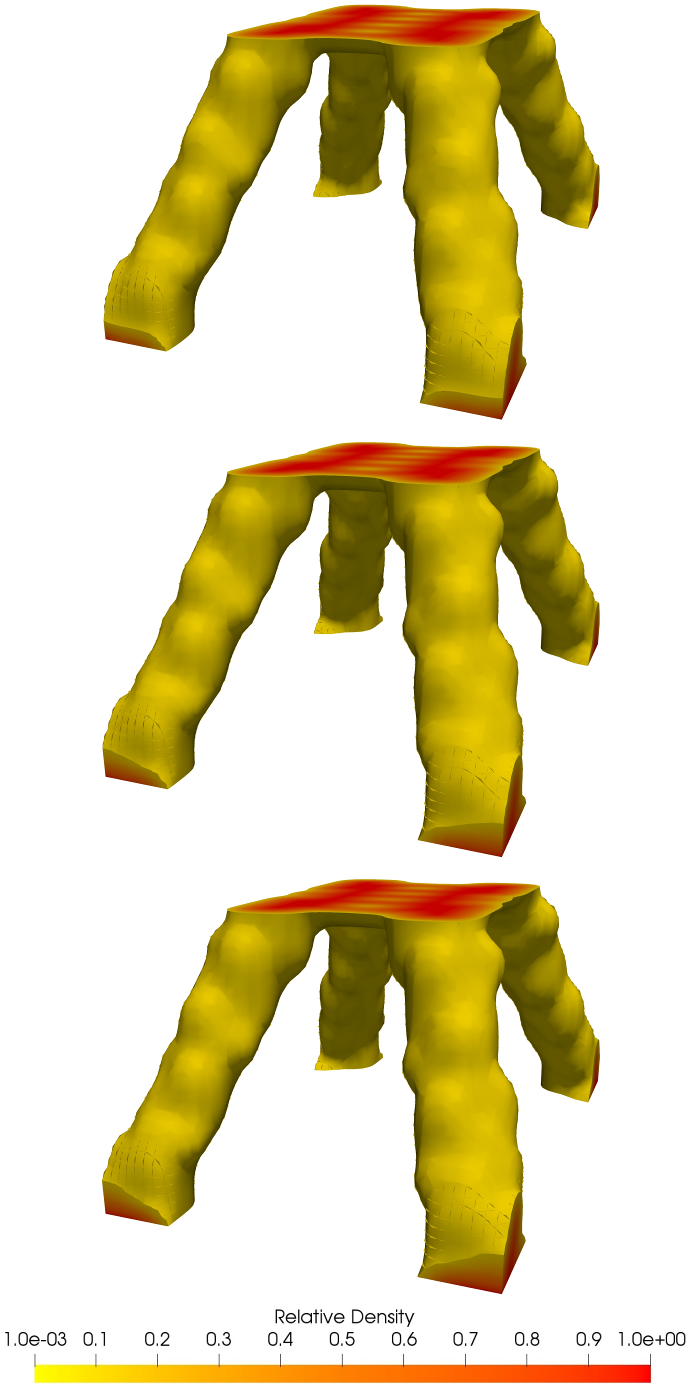
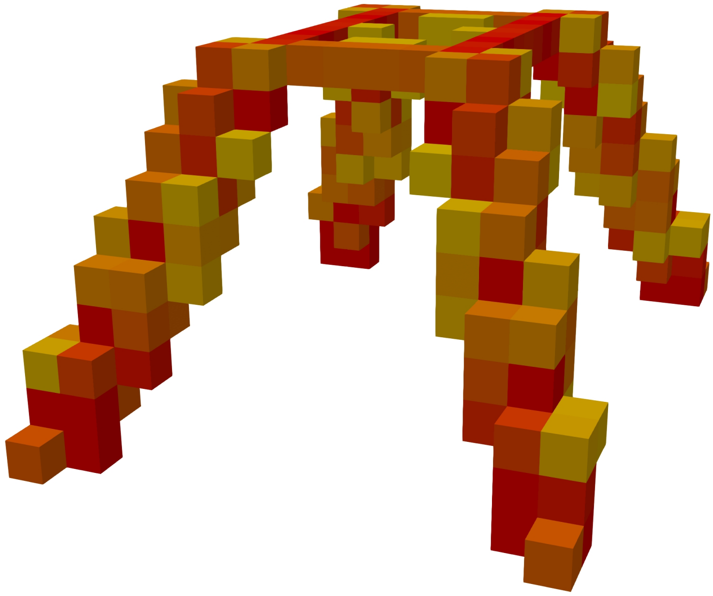
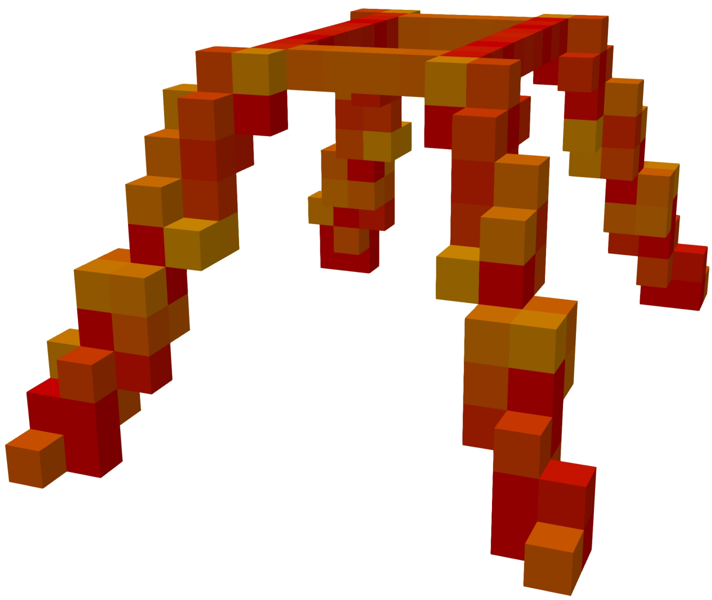


Figure 9.66. Plate with large height and four upper loads: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Isogeometric Analysis [iterations 900, 1200, 1500]



(a) Areas with a relative density greater than or equal to 0.1



(b) Areas with a relative density greater than or equal to 0.3

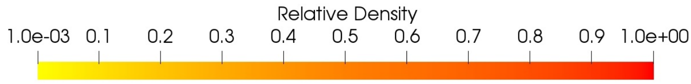
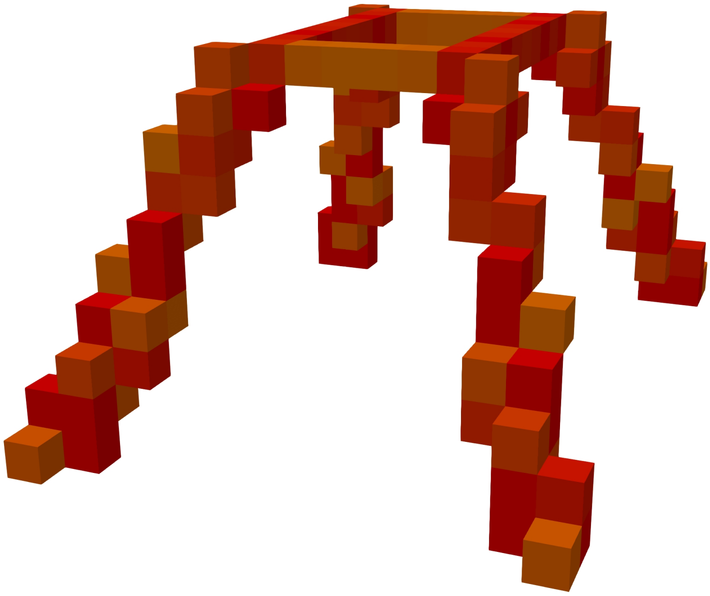
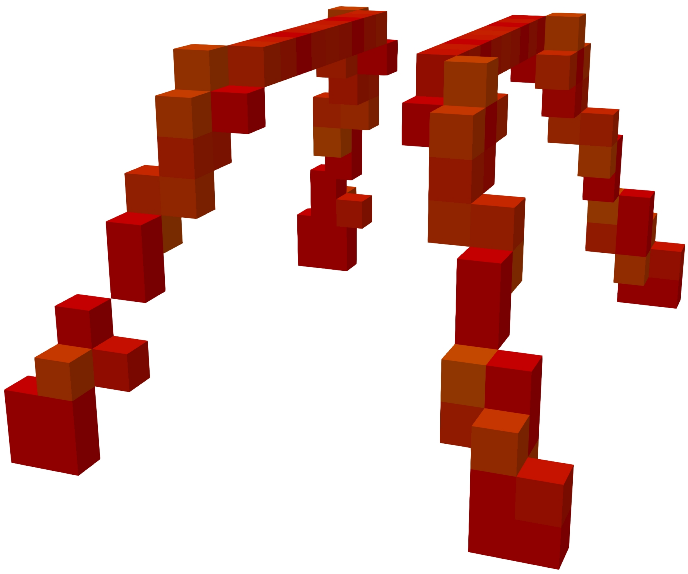


Figure 9.67. Plate with large height and four upper loads: Optimal solution by using the Finite Element Method



(a) **Areas with a relative density greater than or equal to 0.5**



(b) **Areas with a relative density greater than or equal to 0.7**

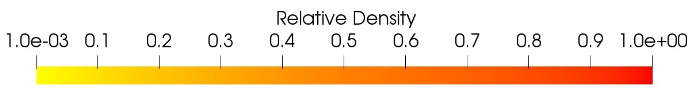
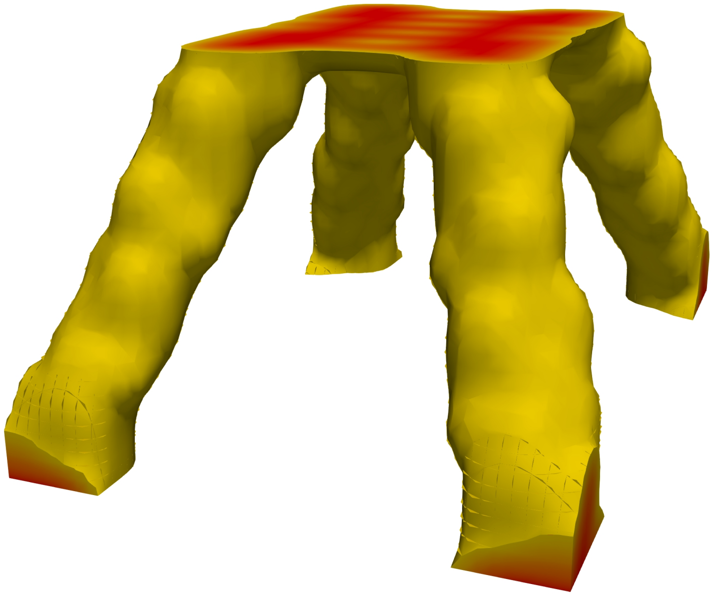


Figure 9.68. Plate with large height and four upper loads: Optimal solution by using the Finite Element Method



(a) Areas with a relative density greater than or equal to 0.1

(b) Areas with a relative density greater than or equal to 0.3

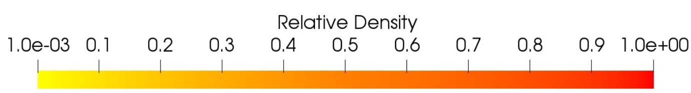


Figure 9.69. Plate with large height and four upper loads: Optimal solution by using the Isogeometric Analysis

(a) **Areas with a relative density greater than or equal to 0.5**

(b) **Areas with a relative density greater than or equal to 0.7**

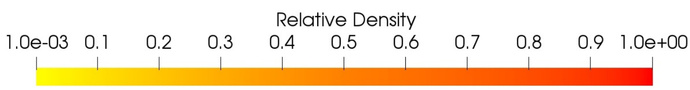
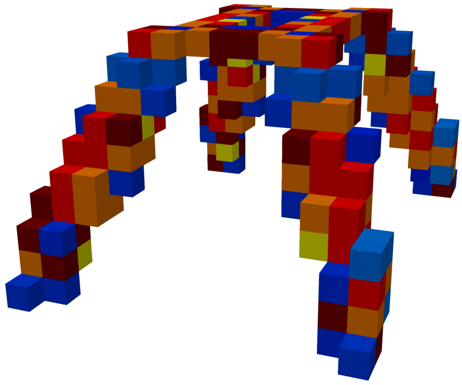
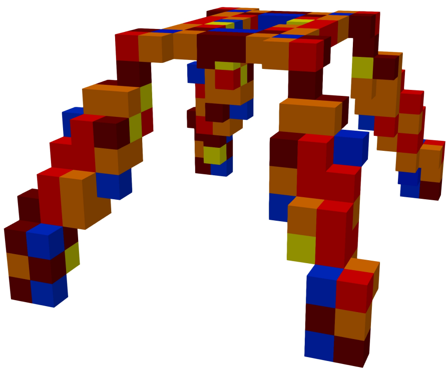


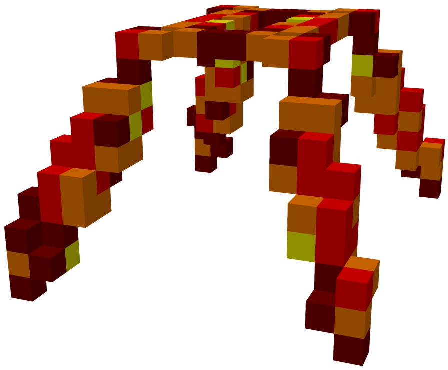
Figure 9.70. Plate with large height and four upper loads: Optimal solution by using the Isogeometric Analysis



(a) Areas with a normalized stress greater than or equal to 0.5



(b) Areas with a normalized stress greater than or equal to 0.8



(c) Areas with a normalized stress greater than or equal to 1

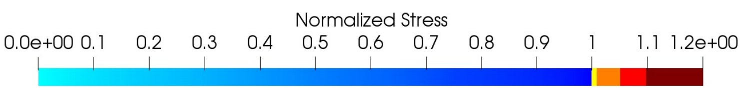
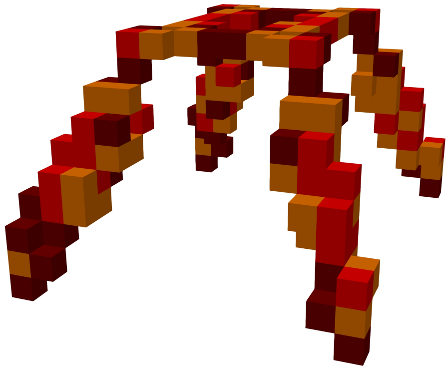


Figure 9.71. Plate with large height and four upper loads: Normalized stress by using the Finite Element Method



(a) Areas with a normalized stress greater than or equal to 1.01

(b) Areas with a normalized stress greater than or equal to 1.05

(c) Areas with a normalized stress greater than or equal to 1.1

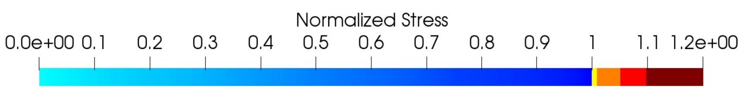
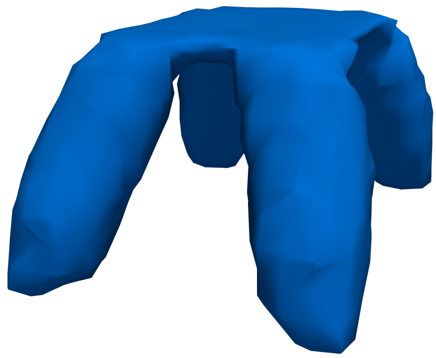
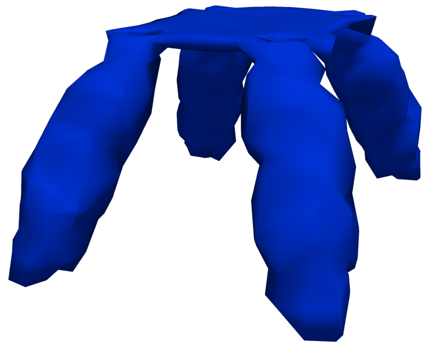


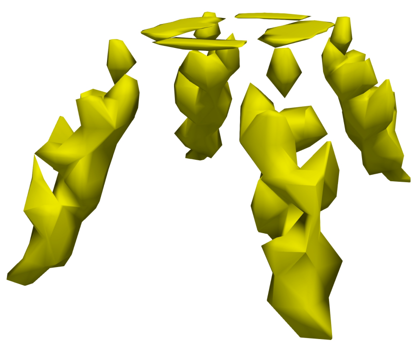
Figure 9.72. Plate with large height and four upper loads: Normalized stress by using the Finite Element Method



(a) Areas with a normalized stress greater than or equal to 0.5



(b) Areas with a normalized stress greater than or equal to 0.8



(c) Areas with a normalized stress greater than or equal to 1

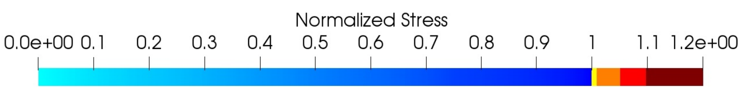
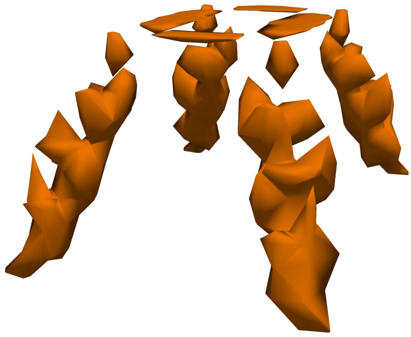
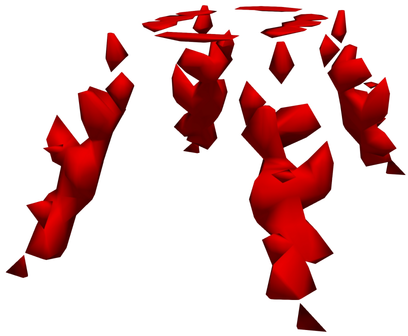


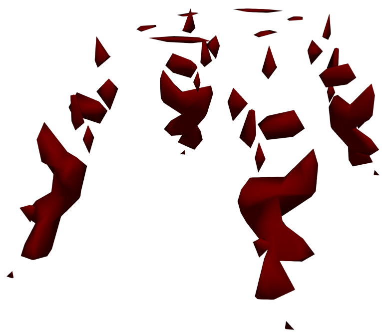
Figure 9.73. Plate with large height and four upper loads: Normalized stress by using the Isogeometric Analysis



(a) Areas with a normalized stress greater than or equal to 1.01



(b) Areas with a normalized stress greater than or equal to 1.05



(c) Areas with a normalized stress greater than or equal to 1.1

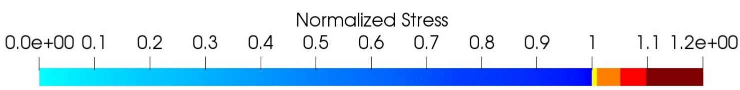


Figure 9.74. Plate with large height and four upper loads: Normalized stress by using the Isogeometric Analysis

	Finite Element Method	Isogeometric Analysis
Number of design variables	2940	4416
Number of constraints	1	1
Number of iterations	1000	1500
Final weight (kg)	132992.8	148628.8
CPU time (h)	20.18	25.93
Time per iteration (s)	72.7	62.3
Structural analysis time / Total time	95.47%	18.78%
Sensitivity analysis time / Total time	4.47%	80.40%

Table 9.29. Plate with large height and four upper loads: General parameters of the problem

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.30. Plate with large height and four upper loads: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-199	0-299
2	0.001	200-399	300-599
3	0.002	400-599	600-899
4	0.003	600-799	900-1199
5	0.005	800-1000	1200-1500

Table 9.31. Plate with large height and four upper loads: Value of the parameters during the resolution of the problem and range of application

Since only one constraint is used to introduce the effect of local stress constraints the memory space required is basically just the needed to store the structural stiffness matrix in both methods. As a result of this and similarly to the two-dimensional problems, the conventional Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis.

Furthermore, the number of iterations of the topology optimization algorithm used to obtain the final solution has been 1000 in case of the conventional Finite Element Method and 1500 in case of the Isogeometric Analysis. However, it is possible to observe in figures 9.63 and 9.66 that there are no considerable differences between the solution obtained for a smaller number of iterations and the final solution. For this

	Finite Element Method	Isogeometric Analysis
Initial moving limits of the design variables	0.005	
Factor of evolution of the moving limits between sets of iterations	0.75	
Number of iterations between two consecutive modifications	100	150

Table 9.32. Plate with large height and four upper loads: Evolution of the moving of the design variables during optimization

reason, it is possible to ensure that the optimal solution can be achieved with a smaller number of iteration. Nevertheless, the use of extra-iterations allows to ensure that the solution has converged to its optimum.

Although the ideal situation would be to stop the algorithm with the convergence of the solution, it is not suitable in this case due to the high non-linearity of the problem, especially in the Damage Constraint.

On the other hand, the final solutions obtained with both methods are equivalent in terms of topology, since their material layout is quite similar. The final solution consists in four bars that connect the points where each external load is applied and the nearest fixed support, and in the upper part of the domain there is a structure that connect the four points where the external loads are applied. This structure supposes the main difference between both methods, since this structure consists in four bars that connect each point where the loads are applied with the two nearest points where another load is applied in case of the conventional Finite Element Method, and this structure consists in a plate placed in the upper border of the domain that connects the four points where the loads are applied in case of the Isogeometric Analysis.

Finally, it is important to remark that the difference regarding the amount of material required to manufacture the optimal solution obtained with both methods is quite small since the difference is only the 12% of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout does not suppose an important difference in the results obtained with respect to its structural weight.

9.4.2. 3D cantilever beam

The second example of this section is a cantilever beam. This example will have a rigid support in three parts of the left border of the domain with null displacements in the three spatial directions. Moreover, a vertical distributed load will be applied in the middle of the lower part of the right edge of the domain. In the same way that the previous example, this example has been also included in this section due to the analogy between it and the cantilever beam previously solved. Although both examples are analogous, it is not possible to solve the problem proposed in this section in the two-dimensional space because the loads and the supports are not contained in the same plane.

The dimensions of the domain used to solve this problem and the position of the external loads applied can be seen in figure 9.75. Furthermore, the structural weight will be considered as a structural load.

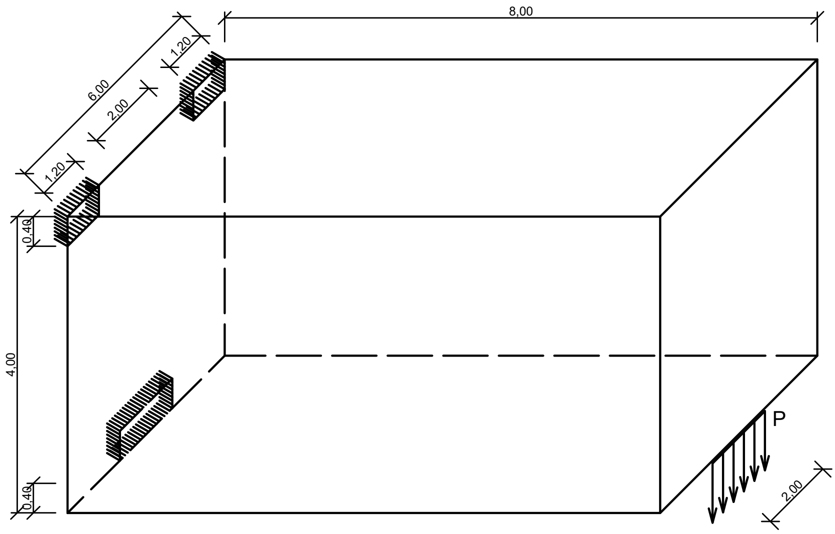


Figure 9.75. 3D cantilever beam: Problem statement. (units - m)

The structural domain will be discretized by means of a regular mesh of 20 x 10 x 15 = 3000 quadratic serendipity elements with 20 nodes in the case of the conventional Finite Element Method and the same number of quadratic knot spans with 27 nodes in the case of the Isogeometric Analysis.

An external distributed load of 625 10^3 kN/m is applied over 5 adjacent elements in the middle of the lower part of the right edge of the domain. The material used for the design of this structure is steel with the same properties of the previous example.

Figures 9.76, 9.77, 9.78, 9.79, 9.80 and 9.81 represent the evolution of the optimal design during the topology optimization process. The images are filtered by removing densities lower than 0.1. Figures 9.82, 9.83, 9.84 and 9.85 show the part of the solution whose relative density is greater than a set of given level-set thresholds in order to better understand and view the three-dimensional optimal solution obtained.

As it can be seen in figures 9.82, 9.83, 9.84 and 9.85, the solutions obtained with both methods are practically equal. The main differences between both solutions correspond to the material layout discretization of each method.

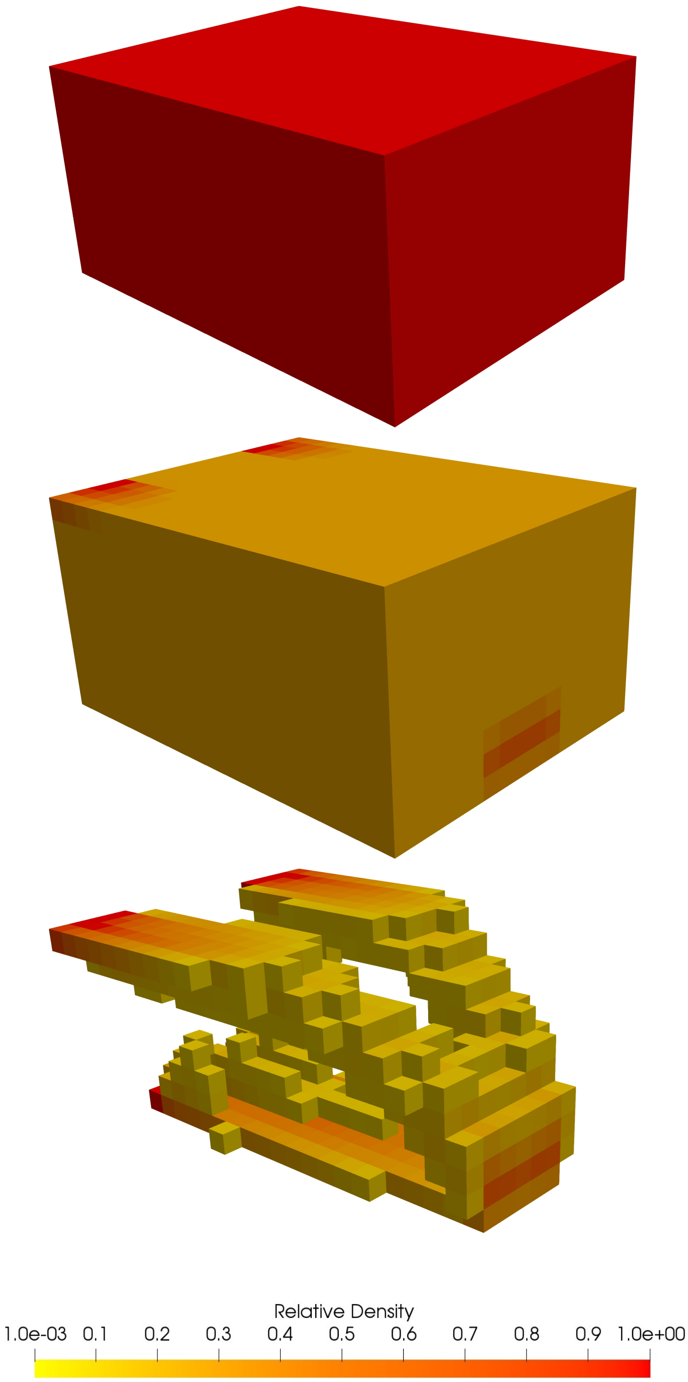


Figure 9.76. 3D cantilever beam: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Finite Element Method [iterations 0, 150, 300]

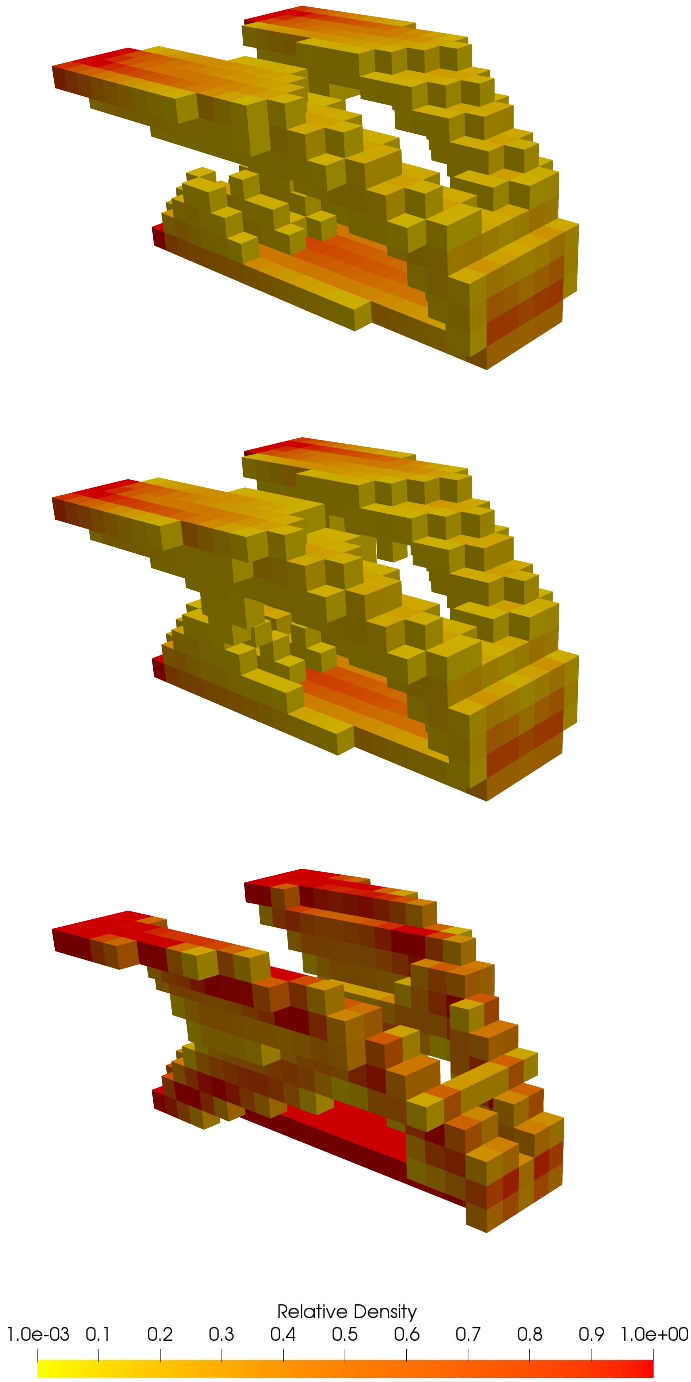


Figure 9.77. 3D cantilever beam: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Finite Element Method [iterations 450, 600, 1200]

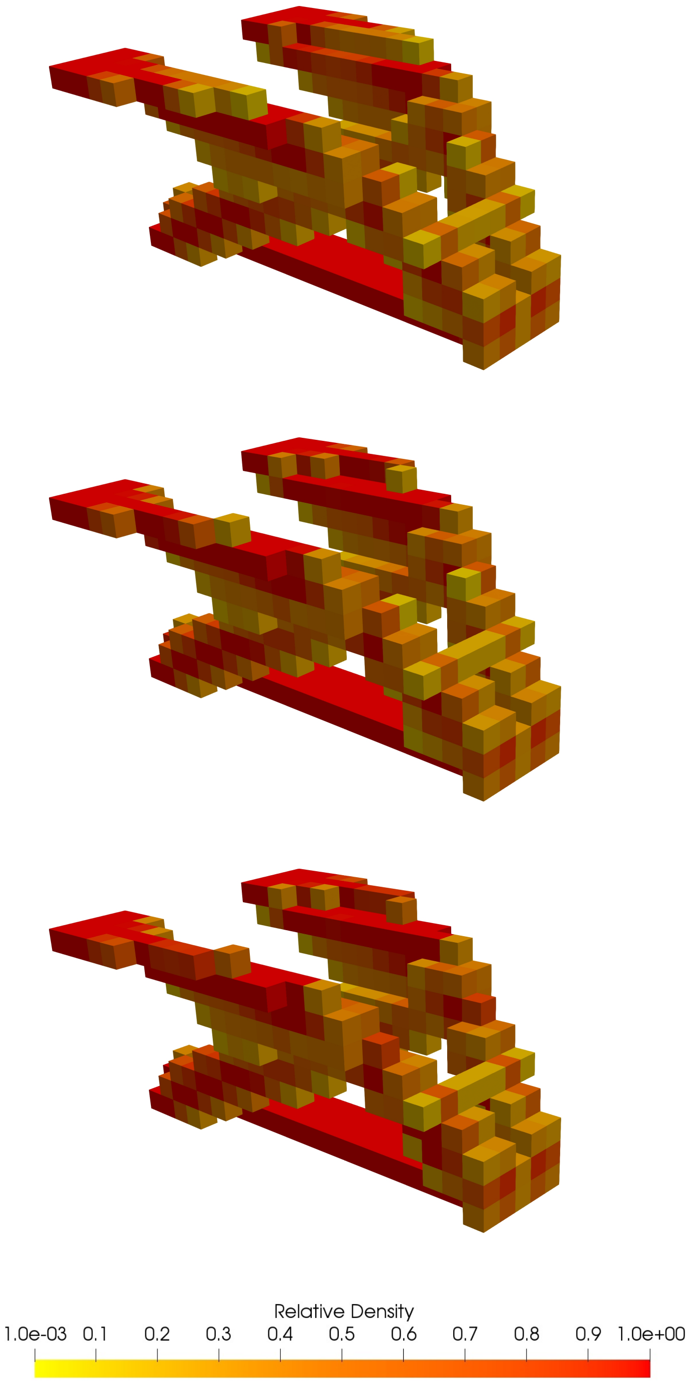


Figure 9.78. 3D cantilever beam: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Finite Element Method [iterations 1800, 2400, 3000]

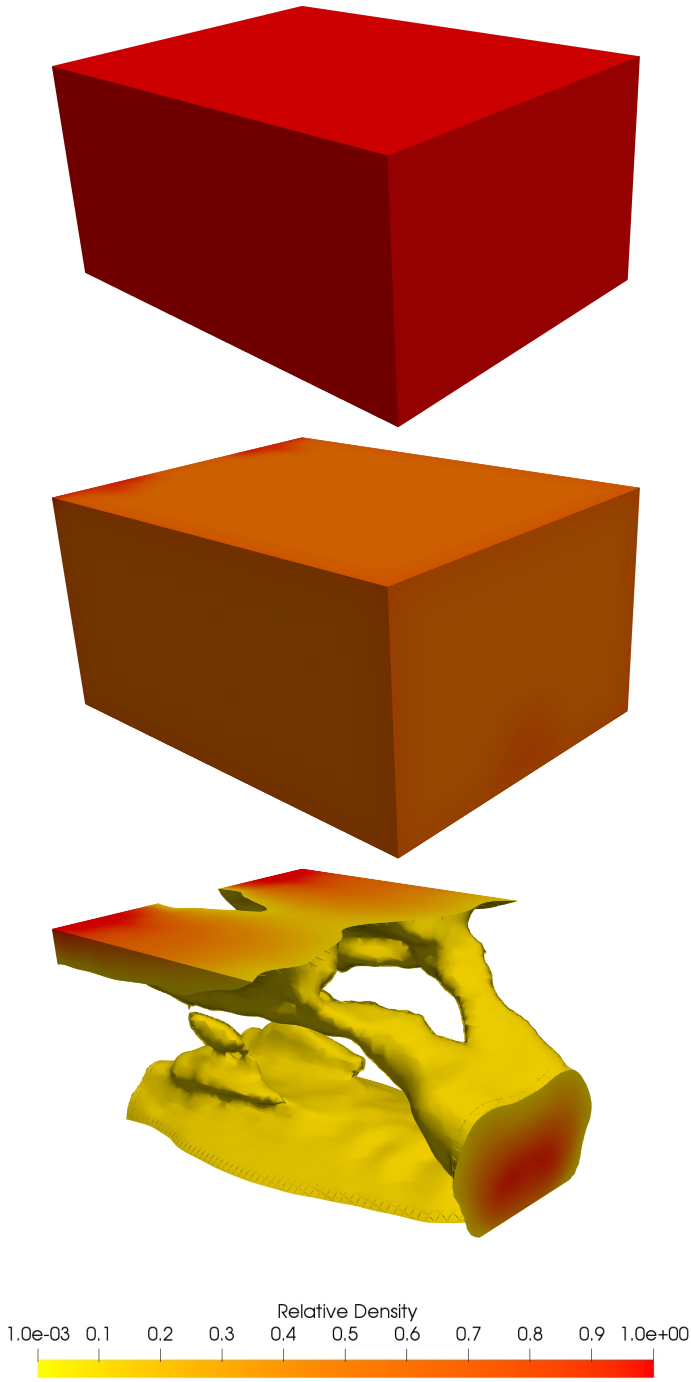


Figure 9.79. 3D cantilever beam: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Isogeometric Analysis [iterations 0, 150, 300]

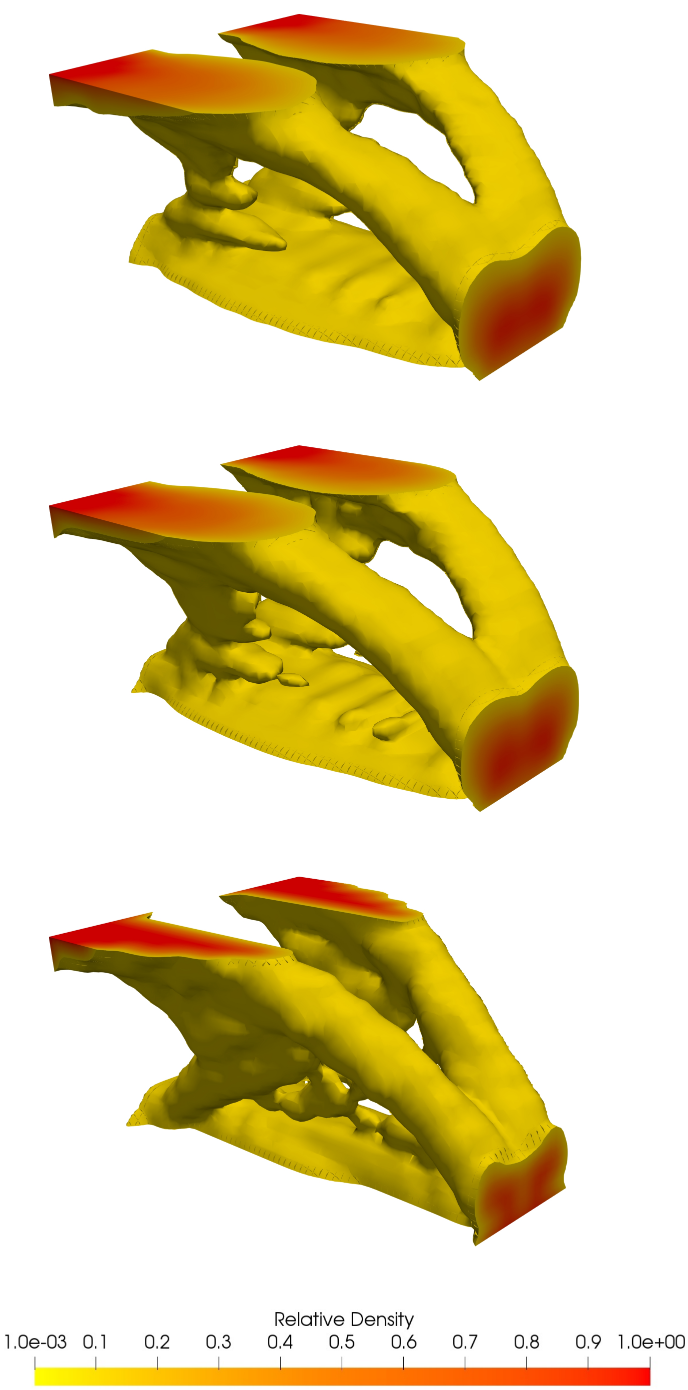


Figure 9.80. 3D cantilever beam: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Isogeometric Analysis [iterations 450, 700, 1400]

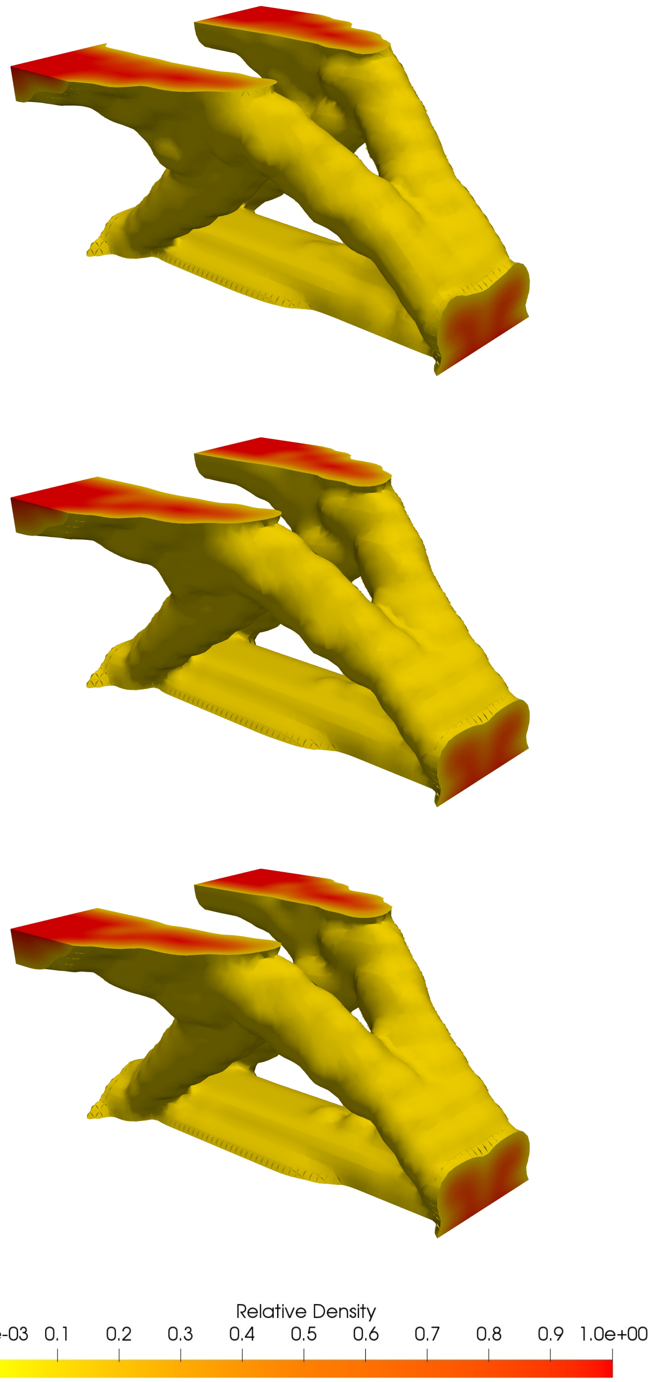
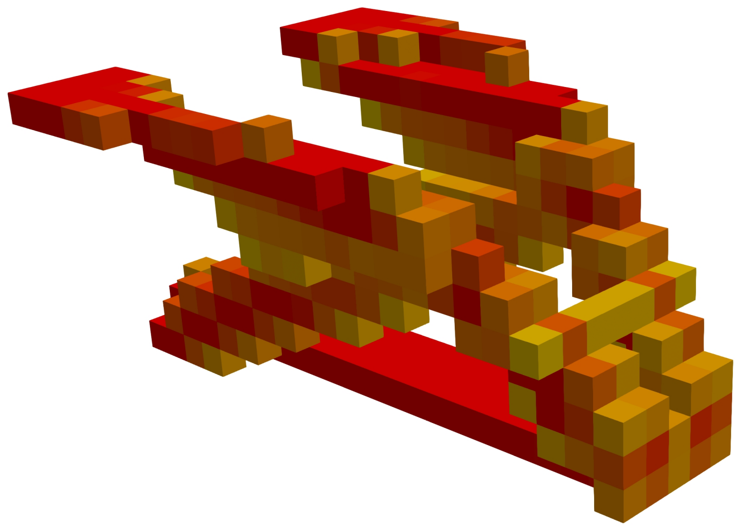
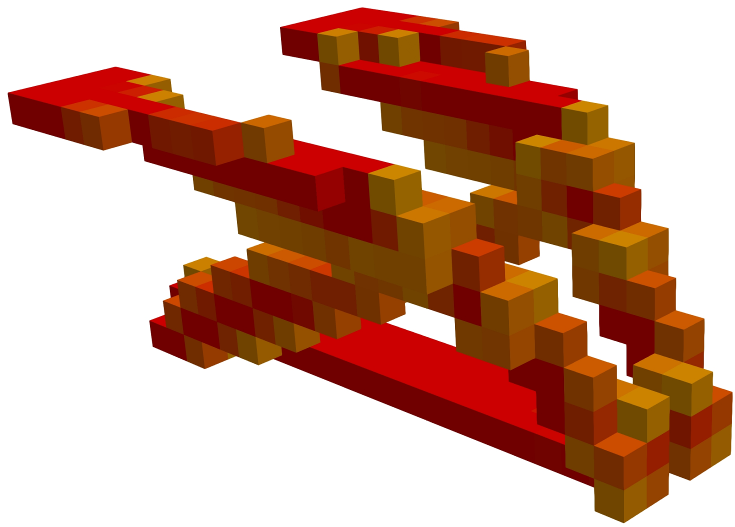


Figure 9.81. 3D cantilever beam: Evolutionary process of the relative density. Plotting level set threshold greater than or equal to 0.1 by using the Isogeometric Analysis [iterations 2100, 2800, 3500]



(a) Areas with a relative density greater than or equal to 0.1



(b) Areas with a relative density greater than or equal to 0.3

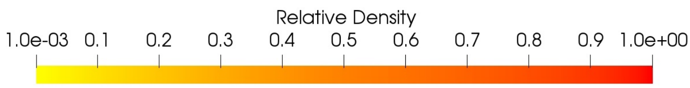
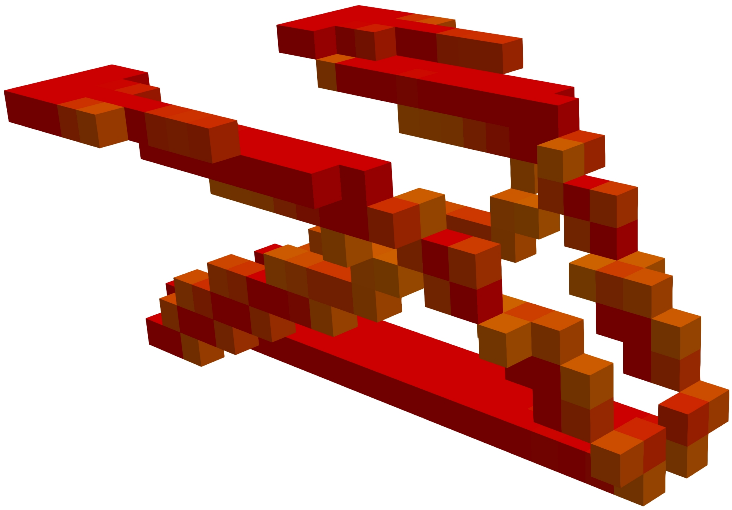
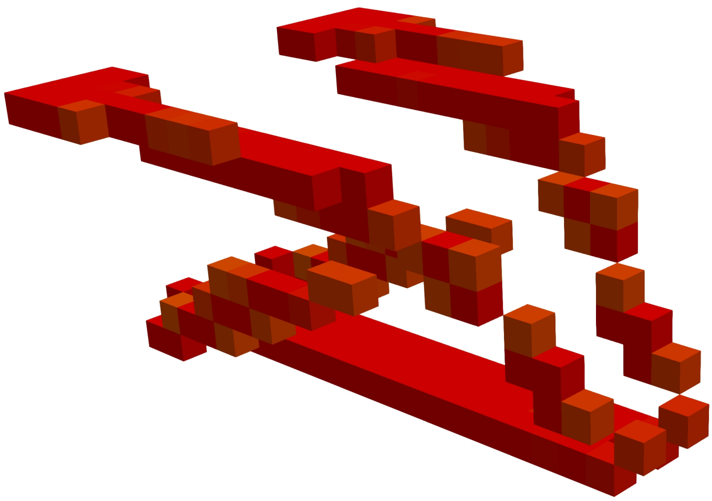


Figure 9.82. 3D cantilever beam: Optimal solution by using the Finite Element Method



(a) Areas with a relative density greater than or equal to 0.5



(b) Areas with a relative density greater than or equal to 0.7

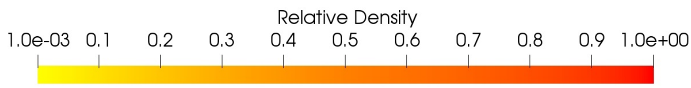
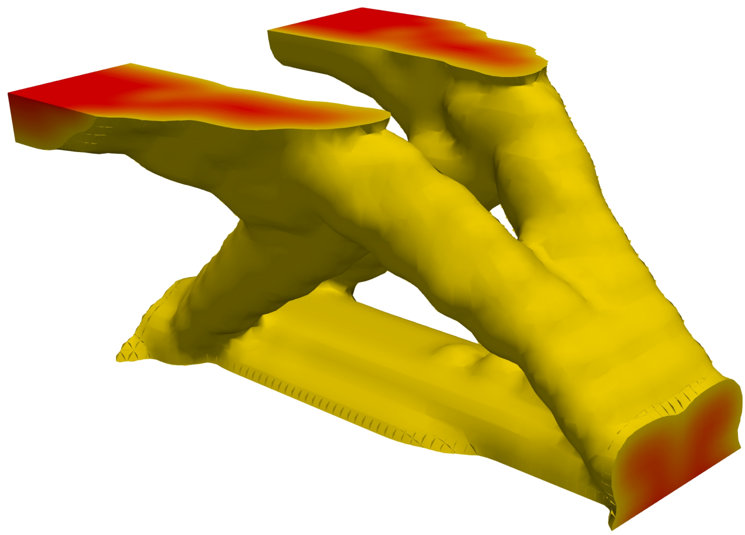
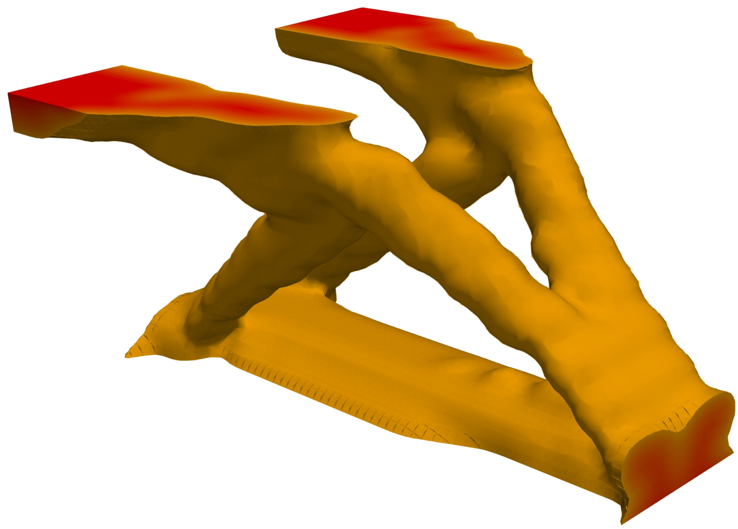


Figure 9.83. 3D cantilever beam: Optimal solution by using the Finite Element Method



(a) Areas with a relative density greater than or equal to 0.1



(b) Areas with a relative density greater than or equal to 0.3

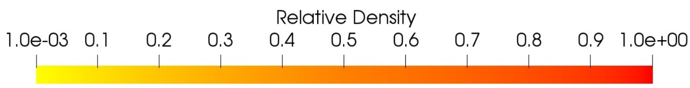
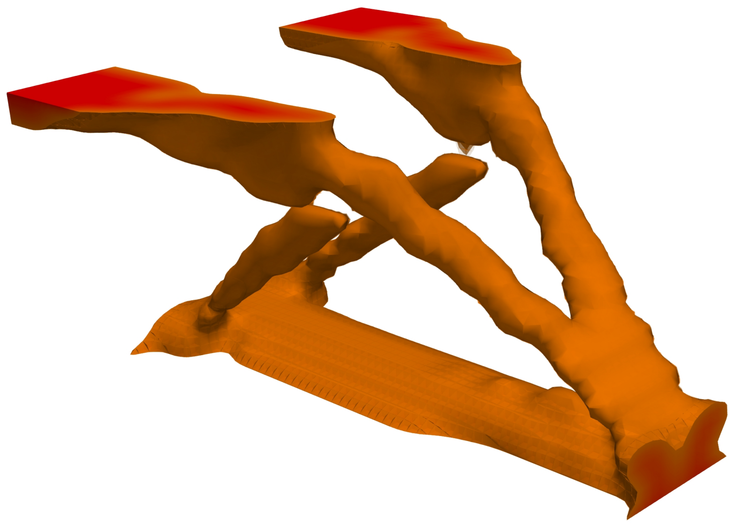
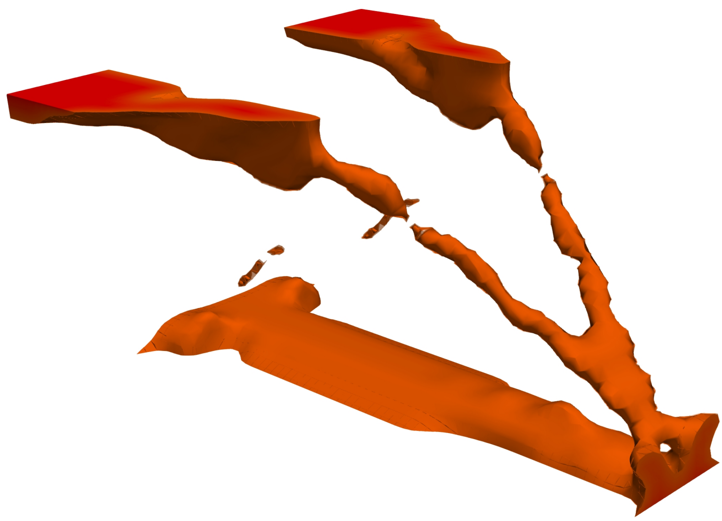


Figure 9.84. 3D cantilever beam: Optimal solution by using the Isogeometric Analysis



(a) Areas with a relative density greater than or equal to 0.5



(b) Areas with a relative density greater than or equal to 0.7

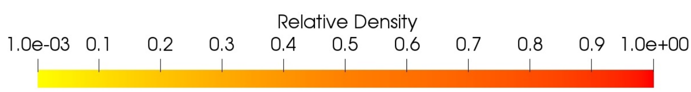
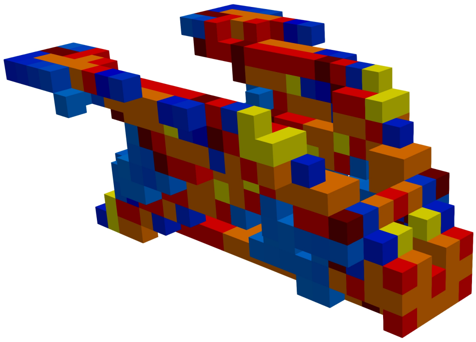
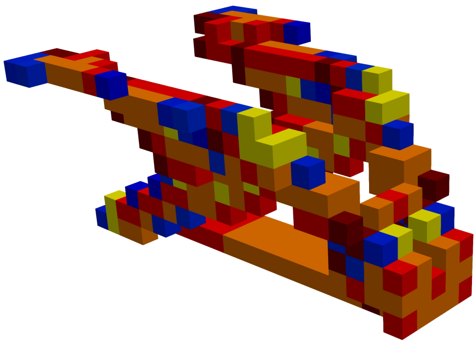


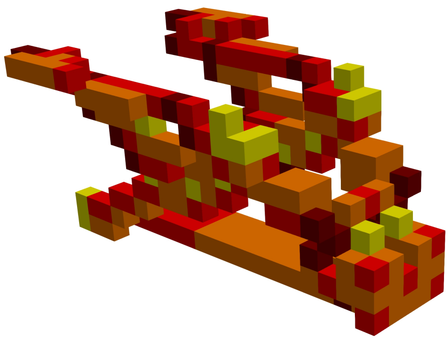
Figure 9.85. 3D cantilever beam: Optimal solution by using the Isogeometric Analysis



(a) Areas with a normalized stress greater than or equal to 0.5



(b) Areas with a normalized stress greater than or equal to 0.8



(c) Areas with a normalized stress greater than or equal to 1

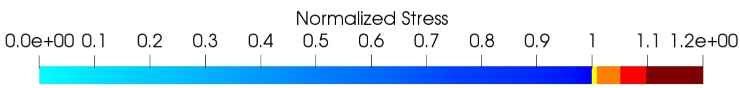
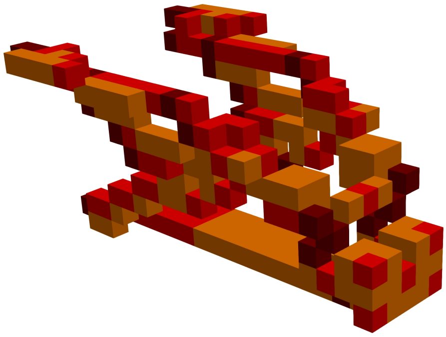
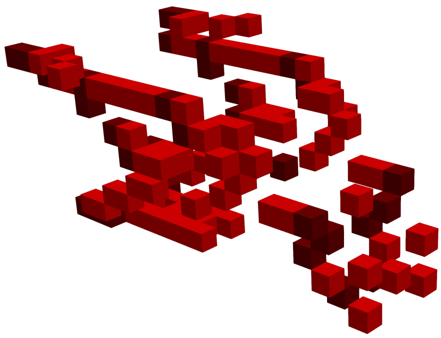


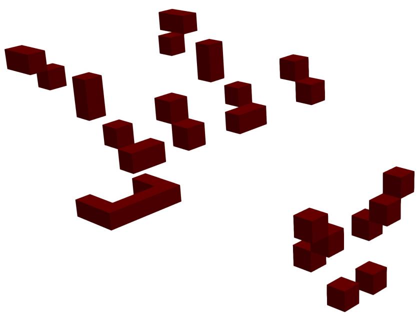
Figure 9.86. 3D cantilever beam: Normalized stress by using the Finite Element Method



(a) Areas with a normalized stress greater than or equal to 1.01



(b) Areas with a normalized stress greater than or equal to 1.05



(c) Areas with a normalized stress greater than or equal to 1.1

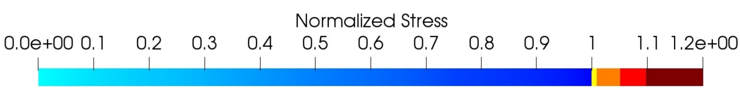
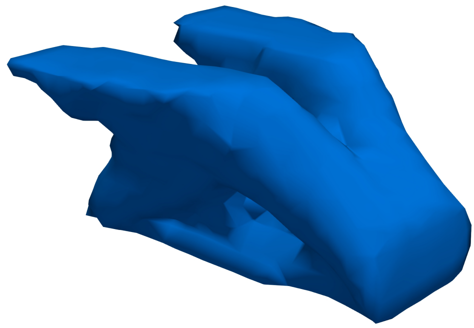
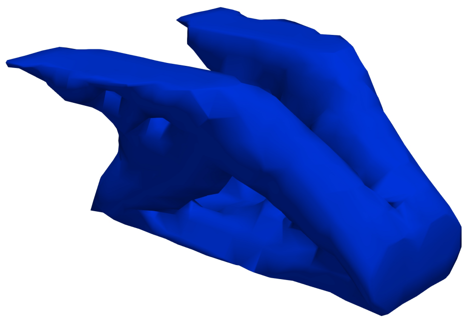


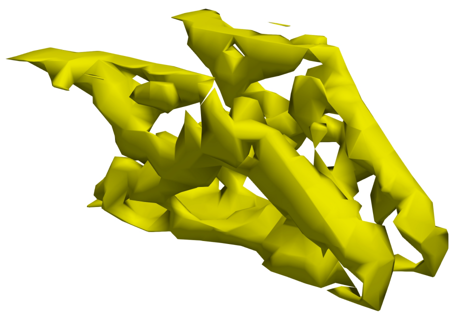
Figure 9.87. 3D cantilever beam: Normalized stress by using the Finite Element Method



(a) Areas with a normalized stress greater than or equal to 0.5



(b) Areas with a normalized stress greater than or equal to 0.8



(c) Areas with a normalized stress greater than or equal to 1

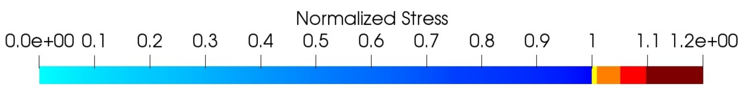
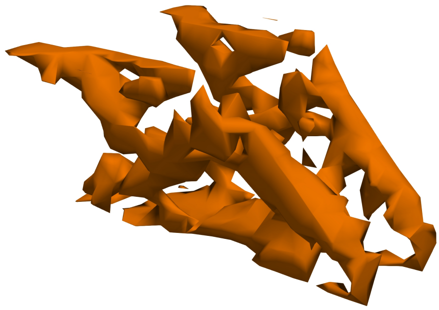


Figure 9.88. 3D cantilever beam: Normalized stress by using the Isogeometric Analysis



(a) Areas with a normalized stress greater than or equal to 1.01

(b) Areas with a normalized stress greater than or equal to 1.05

(c) Areas with a normalized stress greater than or equal to 1.1

Figure 9.89. 3D cantilever beam: Normalized stress by using the Isogeometric Analysis

Figures 9.86, 9.87, 9.88 and 9.89 represent the stress state by means of different level set thresholds for the normalized stress. The value of the normalized stress is equal to the quotient between the stress in each point of the domain and the stress relaxation coefficient times the maximum allowable stress in that point.

Figures 9.87 and 9.89 show that there are regions whose stress value is slightly higher than their maximum. However, this circumstance is due to the relaxation of the damage constraint, and tends to appear in areas with relative density close to its lower limit since the own definition of the damage constraint implies that the stresses can take whatever value if the relative density is equal to its lower limit and the value of the stress relaxation coefficient chosen to solve the topology optimization problem is not big enough with the purpose of counteracting the effect of the relaxation of the damage constraint.

Additionally, the most important characteristic of this example is the existence of symmetry in the final solution obtained between the front and the back part of the structure, considering that the supports are placed in the left part and the external load is applied in the right part.

Tables 9.33, 9.34, 9.35 and 9.36 show the value of the most important parameters introduced in the topology optimization problem. It can be observed regarding the computational requirements that the CPU time required to solve this three-dimensional problem with the Isogeometric Analysis proposed is lower than with the conventional Finite Element Method despite of having to use more iterations to obtain the solution with the Isogeometric Analysis.

	Finite Element Method	Isogeometric Analysis
Number of design variables	3000	4488
Number of constraints	1	1
Number of iterations	3000	3500
Final weight (kg)	128971.2	134050.4
CPU time (h)	63.06	61.86
Time per iteration (s)	75.7	63.7
Structural analysis time / Total time	95.59%	20.19%
Sensitivity analysis time / Total time	4.35%	79.03%

Table 9.33. 3D cantilever beam: General parameters of the problem

Damage coefficient (α)	50
Translation of the original damage function (units) (φ)	0.01
Size of the definition range of the transition function (ε)	0.1

Table 9.34. 3D cantilever beam: Parameters of the damage constraint

Penalty coefficient	Stress relaxation coefficient	Finite Element Method (iterations)	Isogeometric Analysis (iterations)
1	0	0-599	0-699
2	0.001	600-1199	700-1399
3	0.002	1200-1799	1400-2099
4	0.003	1800-2399	2100-2799
5	0.005	2400-3000	2800-3500

Table 9.35. 3D cantilever beam: Value of the parameters during the resolution of the problem and range of application

	Finite Element Method	Isogeometric Analysis
Initial moving limits of the design variables	0.005	
Factor of evolution of the moving limits between sets of iterations	0.75	
Number of iterations between two consecutive modifications	300	350

Table 9.36. 3D cantilever beam: Evolution of the moving of the design variables during optimization

On the other hand, the memory space required is basically the needed to store the structural stiffness matrix in both methods, since just one constraint is used to introduce the effect of local stress constraints. As a result of this, the Finite Element Method will have more computational requirements in terms of memory than the Isogeometric Analysis.

Furthermore, the number of iterations required has been 3000 in case of the conventional Finite Element Method and 3500 in case of the Isogeometric Analysis. However, it is possible to observe in figures 9.78 and 9.81 that there are no considerable differences between the solution obtained for a smaller number of iterations and the final solution. This circumstance allows to ensure that the solution has converged to its optimum.

The use of a linear approximation for the Damage Constraint in the solution of the problem means the non-existence of convergence properly since the Damage Constraint is extremely non-linear. For this reason, the algorithm will not stop because of the convergence of the solution what would be the ideal situation unless a comparison between two different solutions will be established as a way to analyze the convergence.

On the other side, the final solutions obtained with both methods are equivalent in terms of topology, since the global material layout is similar. The final solution of this problem consists in a certain number of bars that connect the supports with the region where the external load is applied and tend to coincide with the isostatic lines.

Nonetheless, it is possible that the number of bars and their positions will be different depending on the method employed what has influence in the material layout discretization. In the conventional Finite Element Method, the value of the relative density at each point of an element is directly related with only one design variable, however, in the Isogeometric Analysis this relationship is more complex, since this value is a combination of design variables.

The use of a combination of design variables to define the material layout discretization with the Isogeometric Analysis means that when the penalty coefficient is introduced in the formulation of the problem the design variables whose value is close to their minimum allowable value tend to pull the nearest design variables to their minimum allowable value unless that the value of these design variables is not close enough to their minimum allowable value. This circumstance tends to happen when the value of most design variables that surround one of them is equal to their minimum allowable value. Consequently, the softer bars of the conventional Finite Element Method, what means a value of the relative density close to its lower limit, tend to disappear in the Isogeometric Analysis.

Finally, it is important to remark that the difference regarding the amount of material required to manufacture the optimal solution obtained with both methods is quite small since the difference is only the 4% of the weight of the lighter solution. Therefore, the use of different formulations to define the material layout will not mean an important difference in the results obtained with respect to its structural weight.

9.5. Conclusions

In this chapter, 9 different structural topology optimization problems have been studied. These problems have been solved by means of the formulations proposed in this thesis in order to test their validity. All the examples have been solved with the two approaches developed in this thesis: the conventional Finite Element Method and the Isogeometric Analysis.

The examples solved in this chapter have been divided in three different groups since they have been included for different reasons. The first group is composed by two-dimensional examples whose optimal solution is known from an analytical point of view. Thus, they are included in order to validate the results obtained comparing them with the analytical solutions.

The second group is composed by practical problems in the engineering field that can be solved in the two-dimensional space, since they have been extensively analyzed in the literature. The last group is composed by three-dimensional examples; these examples have been created using two of the examples previously solved in the two-dimensional space as a way of inspiration. These examples are used to obtain optimal three-dimensional solutions.

As it can be seen in the examples solved in this thesis, the structural topology optimization problem of minimum weight with Damage Constraint provides solutions that are similar to the solutions obtained by means of the use of the stress constraints or stress constraint aggregation techniques previously developed. However, the use of the Damage Constraint makes possible the attainment of solutions with a high spatial definition since it is possible to solve the topology optimization problem with an elevated number of design variables and only one constraint what means an important reduction of the CPU time required to solve the problem. As a consequence, the method proposed in this thesis has been validated to solve the topology optimization problem.

Therefore, it is possible to ensure that the solutions obtained are able to support all the loads applied over them since it has taken account the stresses when the constraint of the structural topology optimization problem has been defined. All the examples analyzed in this thesis has been solved by means of the use of only one damage constraint that consider all the structural stresses. This circumstance is important since the number of constraints used to solve the structural topology optimization problem will have an important effect in the CPU time. On the other side, the quality of the results obtained with only one Damage Constraint is better than with the use of the classical stress aggregation techniques where one or more constraints can be used.

However, the structural analysis model used in the solution of the topology optimization problem presents an inherent problem that is related with the presence of traction and compressive pure states in some parts of the domain since they are not correctly solved due to the existence of endless solutions. This problem can be seen in one of the two-dimensional cases solved in this thesis when the Conventional Finite Element Method is used.

Although this problem can be related with the way to define the relative density in the domain. The problem can be solved if a perimeter penalization is introduced in the formulation of the problem. On the contrary, the Isogeometric Analysis is able to solve this problem with the introduction of only the penalization of intermediate values of relative density since the Isogeometric Analysis can not develop sharp transitions from low densities to high densities. This circumstance is due to the presence of the intermediate values of the relative density takes place in the transitions between low and high densities, in other words, if these transitions are considered as the perimeter of the structure, the penalization of intermediate values of relative density will also try to obtain the structure with minimum perimeter.

Even though the solutions obtained with the two different strategies proposed in this thesis to define the material layout in the domain are topologically equivalent, it is possible to conclude that the Isogeometric Analysis proposed provide better results than the conventional Finite Element Method. This conclusion can be easily explained since the solutions obtained with the Isogeometric Analysis proposed will be more smooth in terms of variability of the relative density than with the conventional Finite Element Method, since the relative density discretization will be C^2 continuous in all the structural domain. The solutions obtained with the Isogeometric Analysis will have much better spatial definition than the obtained with the Conventional Finite Element Method for an equivalent number of unknowns.

On the other hand, the CPU time required to solve the structural topology optimization problem with a large number of divisions of the domain with the Isogeometric Analysis will be lower than with the Finite Element Method in two-dimensional problems and in three-dimensional ones. In one of the three-dimensional problems solved in this chapter the CPU time required with the Isogeometric Analysis is higher than with the Finite Element Method model for an equivalent number of unknowns. However, this circumstance is due to the larger number of iterations required in the solution of the problem with the Isogeometric Analysis, since its time per iteration is lower than with the conventional Finite Element Method. Moreover, the smoothness of the material distribution and the quality of the solution is largely improved by using the Isogeometric Analysis versus the conventional Finite Element Method.

Chapter 10

Conclusions

"The future depends on what you do today"
Mahatma Gandhi, (1869-1948).

10.1. Overview

A comprehensive study of the topology optimization problem of continuous structures has been developed in this thesis. The proposed approach states the problem in terms of minimizing the structural weight with constraints on the maximum allowable stress. Stress constraints are imposed by means of a Damage Constraint formulation. Moreover, this technique can be easily extended to include other types of constraints.

The formulation of the Damage Constraint has been developed as an alternative to the classical stress constraints aggregation techniques. The Damage Constraint is able to avoid the main disadvantage of local formulations stress constraints, that is: the unaffordable growth of the CPU time as the number of constraints is increased, and to improve the results obtained with constraints aggregation techniques.

On the other hand, two different procedures have been used for defining the material layout within the domain in the formulation of the structural topology optimization problem.

The first approach consists in using uniform densities per element. Thus, the material distribution in the domain is discontinuous. In this case, the structural analysis is stated in terms of a Finite Element formulation.

The second approach consists in using an Isogeometric Analysis interpolation type to model the material density distribution. Thus, the material distribution in the domain is continuous. In this case, the structural analysis is stated in terms of an Isogeometric Analysis formulation.

The structural problem is stated and solved considering the linear plane stress hypothesis in the two-dimensional problems and the general linear elastic approach in the three-dimensional problems.

Finally, the optimization problem has been solved by means of a Sequential Linear Programming algorithm (SLP). Special care has been taken to avoid the use of higher order derivatives, since keeping the CPU time as low as possible is one of the objectives of this thesis.

10.2. General conclusions

The most relevant conclusions obtained during the development of this thesis are presented below.

With regards to the formulation used for stating the stress constraints:

- The local stress constraints formulation can provide good results, since this approach ensures that the stress in the selected points will be lower than the maximum allowable value. In return, the computational cost can be extremely high.

- The global and the block aggregation of stress constraints formulations can provide acceptable results, although these approaches do not ensure that the stress will be lower than the maximum allowable value. In return, the computation cost could be much lower.

- Neither the local stress constraints formulation on the one hand, nor the global and the block aggregation of stress constraints formulations on the other hand, allow for efficiently solving three-dimensional problems, since the computational requirements could be unaffordable in the first case and the results could lack the necessary accuracy in the second.

On the contrary, with regards to the use of the Damage Constraint formulation as proposed in this thesis:

- The Damage Constraint formulation can provide good results as a general rule, since the effect of a number of local stresses is quite accurately taken into account.

- The computational cost associated to the use of the Damage Constraint formulation is similar to the computational cost required by the global and the block aggregation of stress constraints formulations, since the number of constraints made explicit in the optimization problem is the same.

- The Damage Constraint formulation ensures that all the structural stresses considered are lower than their maximum allowable value from a theoretical point of view, since the weight of the damaged model generated has to coincide with the original one. However, and due to computational issues a little relaxation of the damage constraint is introduced in its formulation to change the equality constraint into an inequality one. In return, a little infringement in the value of the local stresses is allowed.

- The Damage Constraint formulation can provide solutions that are able to support all the loads applied over the structure if a reduced value of the stress relaxation coefficient is used. This circumstance is important from the engineering point of view.

- The Damage Constraint formulation assures the existence of areas with relative density equal to their lower limit without using a stress relaxation coefficient extremely high, since the weight of these areas are not penalized by the Damage Constraint even if the stresses are higher than their maximum allowable value. The Damage Constraint intends to simulate the particular case of relative density equal to zero where stresses do not properly exist.

- The Damage Constraint formulation provides adequate results for three-dimensional problems in an affordable CPU time in comparison with the previously developed approaches.

A specific and adequate optimization technique has been developed in this thesis to solve the proposed structural topology optimization problem. On the other hand, the analytical formulation of the sensitivity analysis has been specifically developed considering the computational requirements.

Moreover, a coefficient that penalizes the intermediate values of the relative density is introduced in the formulation, since one of the most important issues is that the solutions obtained with the algorithm developed will be easily manufactured.

The structural topology optimization problem has been solved by using two different formulations of the material layout:

- A uniform density per element formulation: The value of the relative density in all the points of each element takes a uniform value. The structural analysis is stated in terms of a Finite Element formulation.

- A material density distribution by means of Isogeometric Interpolation: The value of the relative density in each point of the domain depends on the value of the relative density in a certain set of control points and it is computed by using a B-spline interpolation. The structural analysis is stated in terms of an Isogeometric Analysis formulation.

Different methods for solving the structural analysis have been used depending on the material layout discretization considered since a certain coherence between both parts of the formulation has been required. This circumstance can be easily justified, since the global stiffness matrix used to solve the structural analysis depends on the material layout.

With regards to both formulations, the most important differences between them are related with:

- Design variables: The design variables of the first formulation depict the value of the relative density in each element of the mesh. Conversely, the design variables of the second formulation depict the value of the relative density in the control points. The value of the design variables does not coincide with the value of the relative density in any point with the exception of the interpolatory points.

- Structural analysis: The number of points required to compute the structural analysis with the second formulation is lower than with the first one for the same topology optimization problem. Since the number of points required is proportional to the size of the global stiffness matrix, the CPU time required to compute the structural analysis with the second formulation is also lower than with the first one.

- Sensitivity analysis: The relative density in each point of the domain depends on only one design variable in case of the first formulation. However, the relative density in each point of the domain in case of the second formulation is obtained with a B-spline interpolation of the design variables (relative density in the control points). In other words, the relative density depends on a higher number of design variables. Consequently, the CPU time required to compute the sensitivity analysis of the same topology optimization problem with the first formulation is lower than with the second one.

- Penalization of intermediate values of the relative density: The main objective of the penalization of intermediate values of relative density is to obtain full-void solutions. In the second formulation, this penalization in combination with the high order of continuity in the material distribution produces a similar effect that a perimeter penalization, since the intermediate values of the relative density are placed in the transitions between low and high densities that represent the perimeter of the structure. In other words, if the presence of intermediate values of the relative density is reduced in the domain, the part of the domain that occupies the perimeter is also reduced. On the other hand, the second formulation avoids the appearance of checkerboard patterns since it can not develop a sharp transition from low densities to high densities.

- Spatial definition of the solutions: The solution of the same topology optimization problem with the second formulation provides results with more spatial definition than with the first one. The only way to improve the spatial definition of the solutions obtained with the first formulation is increasing the number of elements, however, the spatial definition of the solutions obtained with the second formulation can be improved by increasing the number of knot span or by increasing the degree of the B-splines used to define the material layout, in both cases the number of control points is also increased.

- Mesh dependency: The mesh dependency phenomena take place by using both formulations, nevertheless, it is less significant in case of using the second formulation as the degree of the B-splines used to define the material layout is increased.

- Size of the problem (number of elements or knot spans): The CPU time required to solve problems with a small number of elements or knot spans is lower with the first formulation. In return, the CPU time required to solve problems with a higher number of elements or knot spans is lower with the second formulation.

10.3. Final conclusions

Once the structural topology optimization problem has been introduced and the approaches and formulations proposed in this thesis have been developed, it is possible to conclude that the use of the Damage Constraint as a way to incorporate the stress constraints is an efficient alternative to the typical approaches used to introduce them, not only in a local way, but also with the global or block aggregation way, regarding both the CPU time and the quality of the results obtained.

Although the solution of the topology optimization problem of minimum weight with Damage constraint has been developed in this thesis for two different formulations, it is possible to conclude that the most efficient method between the developed in this thesis is the material density distributions by means of Isogeometric interpolation, since one of the main objectives of this thesis is the attainment of solutions with high spatial definition.

This conclusion can be justified since the spatial definition of the solution depends on the number of elements or knot spans used in the definition of the problem that is directly proportional to the CPU time required to solve it. Therefore, it is possible to establish that the material density distributions by means of Isogeometric interpolation, is more competitive in terms of CPU time, since it needs less knot spans for the attainment of solutions with the same spatial definition than elements with the uniform density per element formulation. In other words, if the number of elements or knot spans is the same the solutions obtained with the material density distributions by means of Isogeometric interpolation are considerably smoother and have more spatial definition or a better spatial resolution than the solutions obtained with the uniform density per element formulation.

10.4. Prospective research lines

The formulation of minimum weight and stress constraints of the topology optimization problem of continuous structures is a research line widely studied in recent years. This circumstance is due to their high applicability in the engineering field.

The objective function of minimum weight did not mean an important handicap to solve the topology optimization problem. However, the formulation of the stress constraints was the key point in terms of computational requirements. For this reason, the development of different alternatives to incorporate the stress constraints in the topology optimization problem have required an intense research work to be able to solve the topology optimization problem in a reasonable amount of time.

Once this drawback has been overcome by using the Damage Constraint, it is necessary to establish the aspects that should be analyzed in order to improve the performance of the method proposed in this thesis. The most important aspects considered at this moment are:

- The use of the B-spline surfaces or B-spline solids of a higher degree to define the material layout in order to reduce the number of knot spans required in the attainment of solutions with high spatial definition.

- The use of several damage constraints instead of only one, in a similar way to the block aggregation constraint approach, specially for the solution of problems with an extremely higher number of design variables.

- The use of other kinds of stress constraints that will make possible to consider other materials in the definition of the problem with differences between the traction and compression yield stress.

- The use of materials with nonlinear behavior of the material.

- The development of formulations to solve the structural problem with large deformations and large displacements that includes the buckling effect in case of compression.

- The development of new techniques to solve linear systems with sparse matrices more efficiently.

- The use of parallelization techniques in all the algorithms but mainly in the structural analysis and the sensitivity analysis in order to reduce the CPU time.

- The use of other kinds of objective functions in order to increase the scope of application of the Damage Constraint approach.

- The use of different types of coordinates such as polar, spherical or cylindrical in order to solve problems whose geometry will be easily definable with them.

- The development of the Multiregion approach by considering multiple contacts for the same region with the previous ones in the geometrical definition of the topology optimization problem to solve problems that have holes in their geometry, since only one contact with the regions previously defined has been considered in this thesis.

- The development of the Multiregion approach to the three-dimensional space to solve problems with more complex geometries.

- The study of the definition of non-rectangular or non-prismatic geometries in the Cartesian space with the Isogeometric Analysis in the two-dimensional and three-dimensional space respectively to solve whatever kind of topology optimization problem regardless of its geometry.

Appendix A

Resumen extendido en Español

"Una huelga de intelectuales, que es un supuesto improbable, paralizaría la marcha del mundo"

Camilo José Cela, (1916-2002).

Introducción

Desde que Bendsøe y Kikuchi establecieran las bases de la Optimización Topológica en 1988, múltiples contribuciones han sido desarrolladas en dicho campo. Esas contribuciones tuvieron como objeto incrementar el ámbito de aplicación de la optimización topológica mediante la formulación de diversos problemas que pudiesen ser resueltos mediante la aplicación de sus principios y bases. Eso dio lugar a que la optimización topológica pueda ser empleada en campos tan diversos como la ingeniería o la medicina.

En lo que respecta a la optimización topológica de estructuras, una de las que más interés suscita en el campo de la ingeniería civil, el objetivo principal es la distribución de material en un cierto dominio con el objeto de maximizar o minimizar una cierta característica del elemento que se está diseñando y garantizar a su vez el cumplimiento de una serie de condiciones. Las primeras formulaciones del problema de optimización topológica de estructuras buscaban obtener el diseño de mayor rigidez limitando la cantidad de material empleada, estas suscitaron un enorme interés, siendo actualmente las más utilizadas a pesar de que no era posible asegurar que los elementos diseñados fuesen capaces de desarrollar la función para la cual fueron concebidos. Eso era debido fundamentalmente a que las cargas a las que estaban sometidos podían provocar tensiones inasumibles que ocasionaran la rotura del elemento.

A raíz de eso se desarrollaron formulaciones en las que las tensiones o los desplazamientos se incorporaban como restricciones en el problema de optimización, asegurando así que dichos elementos desempeñasen la función para la cual fueron diseñados. Por otro lado, se desarrolló el planteamiento de mínimo peso, donde el objetivo principal era diseñar estructuras que con la menor cantidad de material posible pudiesen desarrollar su función, en otras palabras, que sean capaces de soportar todas las cargas a las que van a estar sometidas durante su vida útil. Esto último estaba garantizado a través de la incorporación de las restricciones en tensión.

Las formulaciones de mínimo peso con restricciones en tensión suscitaron inicialmente un gran interés debido a su aplicabilidad. Sin embargo, la necesidad de obtener soluciones con alta definición espacial significó a su vez la necesidad de disponer de un mayor número de restricciones en la definición del problema. Tras lo cual y con el objeto de reducir los tiempos de cálculo se hizo necesario el desarrollo de nuevas formas de incorporar las restricciones de tensión en el problema de optimización topológica.

Fue así como se desarrollaron los planteamientos de restricción en tensión de tipo global y reducción por bloques de restricciones de tipo local, donde una única restricción tiene en cuenta el efecto de un conjunto o de la totalidad de los puntos en los que se comprueba la tensión considerados en la formulación del problema. Sin embargo y debido a la forma en la que las restricciones eran formuladas no era posible garantizar que todas las tensiones fueran inferiores a su máximo valor permitido en los resultados obtenidos con esta metodología.

Por otra parte, el otro aspecto relevante en la formulación del problema de optimización topológica de estructuras es la forma en la que se define la distribución del material en el dominio. En las primeras formulaciones el dominio era dividido en un cierto número de regiones en las cuales podía haber o no material. En otras palabras, se requería el uso de variables de diseño discretas. Posteriormente, se desarrolló la posibilidad de utilizar variables de diseño continuas en la formulación del problema, lo que permitió la utilización de algoritmos de optimización en la resolución del mismo mejorando a su vez la calidad de los resultados obtenidos, al no ser necesario llenar completamente cada una de las regiones del dominio, las cuales tenían una densidad relativa, grado de ocupación de la región, constante.

Más adelante y con objeto de mejorar la calidad de los resultados obtenidos y evitar la dependencia de la solución con respecto a la forma de dividir el dominio, se desarrollaron formulaciones en las que la densidad relativa en cada punto del dominio dependía del valor de esta en un cierto conjunto de puntos de control o era directamente calculada a través de una función matemática que dependía de la posición de cada punto dentro del dominio de definición del problema.

Por tanto, es necesaria una formulación del problema de optimización topológica de estructuras de mínimo peso que permita asegurar que todas las tensiones estructurales son inferiores a un cierto valor máximo mediante el uso de un número reducido de restricciones en comparación al número de variables de diseño empleadas en la definición de la distribución de material. A su vez también se pretenden obtener resultados con una alta definición espacial mediante el empleo del menor número de variables de diseño posibles.

Y ese es el objetivo de esta tesis. Desarrollar una metodología de optimización topológica estructural que permita la obtención de las estructuras de mínimo peso que sean capaces de soportar todas las cargas a las que van a estar sometidas y que proporcione soluciones con alta definición espacial mediante el uso de un número razonable de variables de diseño. Por todo ello, los objetivos específicos de la tesis son:

- Incorporar el Análisis Isogeométrico en la formulación del problema de optimización topológica de estructuras como alternativa al Método de los Elementos Finitos.
- Desarrollar la formulación del problema de optimización topológica con una única restricción que permita considerar el efecto de un gran número de tensiones de una forma efectiva.
- Extender el planteamiento del problema de optimización topológica de estructuras del espacio bidimensional al espacio tridimensional.
- Lograr que el algoritmo desarrollado permita resolver el problema de optimización topológica en una cantidad de tiempo razonable.
- Resolver diferentes problemas de optimización topológica de estructuras no solo en el espacio bidimensional sino también en el espacio tridimensional.

Análisis estructural

El análisis estructural del elemento a optimizar se calcula haciendo uso de la formulación estándar del Método de los Elementos Finitos para estructuras continuas, dada la naturaleza continua de las tensiones y de los desplazamientos estructurales. Para emplear esta formulación se han considerado las siguientes hipótesis: pequeños desplazamientos, pequeñas deformaciones y material de comportamiento elástico lineal. Por otra parte, la utilización del Análisis Isogeométrico no supone una dificultad inherente dado que dicho método es derivado del Método de los Elementos Finitos.

Sin embargo, será necesario tener en cuenta la influencia de las variables de diseño en la formulación del análisis estructural, puesto que las variables de diseño son usadas para la definición de la distribución de material en el dominio. Por esa razón, será necesario incorporar las variables de diseño en la formulación de la matriz de rigidez.

Por otro lado, todos los ejemplos bidimensionales planteados en esta tesis serán considerados casos de tensión plana. Siendo por tanto necesario incorporar todos los principios de la tensión plana en la formulación del análisis estructural.

Para el cálculo del análisis estructural además de las cargas externas que serán aplicadas sobre la estructura a optimizar también será necesario incorporar el valor del peso propio de dicha estructura, valor que de igual forma que la matriz de rigidez vendrá condicionada por el valor de las variables de diseño. Finalmente, será necesario también incorporar el efecto de las condiciones de contorno usadas en la definición del problema generalmente asociadas a ausencia de desplazamientos en alguna parte del dominio.

Una vez que el análisis estructural ha sido calculado se obtiene como resultado el valor de los desplazamientos en cualquier punto del dominio y el valor de las reacciones en aquellos puntos en los que los desplazamientos han sido coartados. Por otra parte, con el valor de los desplazamientos en cualquier punto del dominio es posible calcular el valor de las tensiones en cualquier punto. Este es el principal objetivo del análisis estructural desarrollado, puesto que las tensiones serán utilizadas para formular la restricción del problema de optimización topológica.

El análisis estructural representa la parte más costosa computacionalmente para una de las formulaciones propuestas en la definición de la distribución de material en el dominio: densidad uniforme por elemento. Sin embargo, podría ser posible reducir el coste computacional si se incorporan técnicas más eficientes en lo que respecta al almacenamiento de matrices o renumeración de mallas.

Planteamiento del problema de optimización

El planteamiento de cualquier problema de optimización matemática requiere de la definición de tres aspectos básicos: la función objetivo, las restricciones y las variables de diseño. Estas últimas son los parámetros que se modifican durante la resolución del problema.

En el problema de optimización topológica de estructuras formulado en esta tesis la función objetivo se corresponde con el peso estructural, el cual se busca minimizar con el objetivo de reducir los costes de producción y el impacto ambiental que supone la fabricación de un cierto componente. En este caso, la relación entre la función objetivo y las variables de diseño es evidente puesto que estas últimas se usan para definir la distribución de material en el dominio.

Además, y con el objeto de obtener soluciones vacío-lleno en las que la mayoría de sus variables de diseño sean iguales a sus límites inferior o superior, lo cual proporciona soluciones más fácilmente manufacturables, se procede a introducir un coeficiente de penalización de los valores intermedios de densidad relativa.

Por otra parte, todo problema de optimización requiere de la existencia de restricciones. En el caso del problema de optimización planteado en esta tesis, se buscará asegurar que las estructuras diseñadas sean capaces de desempeñar su función sin sufrir problema alguno. Por esa razón será necesario realizar la comprobación del valor de las tensiones en el dominio estructural. Para tal fin se procede a introducir únicamente una restricción basada en el ”Damage Approach” (enfoque del daño) en el problema de optimización topológica.

Dicha restricción será formulada a través de la comparación entre dos modelos: uno de ellos es el original y el otro es un modelo dañado en el cual una de las características del modelo original es perturbada cuando el valor de las tensiones supera su máximo valor permitido. En este caso se procede a incrementar el valor del peso estructural en las regiones del dominio en las que las tensiones superen su máximo valor permitido.

Por otra parte, el incremento del peso estructural será mayor a medida que el grado de superación de la tensión máxima aumente.

Por todo ello, la restricción de daño introducida en el problema de optimización topológica consistirá en una comparación del peso total de la estructura de ambos modelos. Dado que la restricción de daño es originalmente una restricción de igualdad y por lo tanto se activará cuando el peso de ambos modelos sea diferente, será necesario introducir una relajación de esta con el objeto de convertirla en una restricción de desigualdad.

Por otro lado, el análisis de sensibilidad de la restricción no proporcionará ninguna información hasta la frontera entre la región factible y no factible al ser el peso de ambos modelos (original y dañado) igual en toda la región factible, en otras palabras, el valor de la restricción de daño va a ser constante y en consecuencia sus derivadas nulas. Por este motivo, es necesario introducir una pequeña modificación en la definición de la restricción de daño cuando el valor de la tensión supere un cierto porcentaje de la máxima tensión permitida que introduzca cambios entre el peso del modelo original y el modelo dañado.

Sin embargo, la magnitud de dichas modificaciones se podrá considerar despreciable en términos de modificación del peso estructural pero valiosa a la hora de realizar el análisis de sensibilidad puesto que se dispondrá de información de cara a la toma de decisiones en las proximidades de la frontera entre la región factible y no factible.

Además, dado que el material empleado para la definición de la estructura es acero, el criterio utilizado en la formulación de las restricciones de tensión que serán consideradas en la generación del modelo dañado es el de Von Mises.

Por último, las variables de diseño del problema de optimización topológica se utilizarán para definir la distribución del material en el dominio. Para tal fin se proponen dos planteamientos alternativos: uno basado en los planteamientos clásicos del Método de los Elementos Finitos y el otro mediante la aplicación de las bases del Análisis Isogeométrico.

Con respecto al primer método el valor de la densidad relativa toma habitualmente un valor constante en cada una de las divisiones del dominio y en consecuencia el número de variables de diseño es igual al número de particiones del dominio. Por el contrario, en el segundo método el valor de la densidad relativa en cualquier punto del dominio tiene que ser calculado por medio de una combinación lineal de los valores de la densidad relativa en los puntos de control multiplicado por el valor de la función de forma de dicho punto de control en el punto donde se quiere obtener el valor de la densidad relativa. Los valores de la densidad relativa en los puntos de control son las variables de diseño del problema de optimización por lo que el número de variables de diseño es igual al número de puntos de control, valor que depende de las características de definición del modelo de Análisis Isogeométrico considerado.

Por otra parte, quedan por definir los valores límite que pueden tomar las variables de diseño. Mientras que el límite superior de la variable de diseño es igual a la unidad, lo que indica ausencia de porosidad, el límite inferior debería ser igual a

0 lo que significaría ausencia de material. Sin embargo, esto no es posible por cuestiones numéricas. En el caso de permitir la aparición de densidades relativas nulas, la matriz de rigidez obtenida en el análisis estructural sería singular y en consecuencia dicho análisis no tendría solución. Por este motivo es necesario establecer un límite inferior de las variables de diseño ligeramente superior a 0. Finalmente, para permitir la aparición de áreas cuya densidad relativa sea igual a su límite inferior se introduce una relajación de las tensiones.

Metodología de optimización y análisis de sensibilidad

Como norma general la propia definición del problema de optimización determina en gran medida la elección del algoritmo matemático de optimización más adecuado para la resolución del problema. El problema de optimización que se pretende resolver en esta tesis consta de una única función objetivo, el peso de la estructura; una única restricción, la restricción de daño; y con variables de diseño continuas en su dominio al ser usadas para definir una variable continua, la densidad relativa del material en el dominio de definición de la estructura.

El tipo de problema a resolver en esta tesis carece de solución analítica directa. Por esa razón es necesario plantear la reducción del problema a una sucesión de problemas más sencillos. En otras palabras, el problema de optimización se resuelve por medio de un proceso iterativo. En esta tesis se ha optado por el mismo método de reducción empleado típicamente en la resolución de los problemas de optimización topológica: Programación Lineal Secuencial (SLP).

Aunque el problema de optimización es no lineal tanto en lo que respecta a la función objetivo como a la restricción de daño, se aproxima a una secuencia de problemas lineales haciendo uso para ello de la información obtenida en el análisis de sensibilidad de primer orden. Por lo tanto, en cada iteración del proceso de optimización se resolverá un problema linealizado mediante la aplicación del algoritmo Simplex.

Sin embargo, es importante tener en cuenta que la linealización del problema implica una pérdida de información con respecto al problema de optimización original, lo que a su vez produce una modificación del problema de optimización a resolver, que tendrá un dominio factible distinto. Por este motivo y con el objetivo de mitigar las diferencias entre el problema original y el linealizado se introduce una limitación adicional en el problema modificado relacionada con la máxima modificación permitida de cada una de las variables de diseño en cada iteración del proceso de optimización. Esta estrategia recibe el nombre de límites móviles, los cuales deben ser actualizados en cada iteración teniendo en cuenta no sólo la máxima modificación permitida sino también el valor de los límites laterales impuestos en la definición del problema de optimización topológica: las densidades relativas mínima y máxima.

Por todo ello, para la transformación del problema de optimización original en el problema linealizado se requiere únicamente del análisis de sensibilidad de primer orden de la función objetivo y de la restricción de daño. Mientras que el análisis de sensibilidad de la función objetivo puede ser calculado directamente, el análisis de sensibilidad de la restricción de daño tiene que ser calculado haciendo uso de las técnicas de diferenciación analítica. En este caso al ser el número de restricciones impuestas mucho menor que el número de variables de diseño, puesto que sólo se ha definido una única restricción de daño, se procede a emplear el Método de la Variable Adjunta.

El análisis de sensibilidad representa la parte más costosa computacionalmente para una de las formulaciones propuestas en la definición de la distribución de material en el dominio: la distribución de material por medio de una interpolación isogeométrica. Sin embargo, sería posible reducir el coste computacional si se implementan las técnicas de paralelización en el cálculo.

Resultados, conclusiones y futuras líneas de investigación

La metodología de optimización topológica de estructuras de mínimo peso con restricción de daño desarrollada en esta tesis se ha aplicado a diferentes ejemplos tanto para la resolución de problemas bidimensionales en tensión plana como para la resolución de problemas tridimensionales. Con este método se han obtenido por un lado reducciones de tiempo de cálculo importantes en comparación con los métodos que incorporaban directamente las restricciones de tensión en el problema de optimización topológica, y por otro lado los resultados obtenidos presentan una elevada definición espacial especialmente en el caso del Análisis Isogeométrico. Esto ha sido posible mediante el incremento del número de variables de diseño usadas en la definición de la distribución del material en el dominio. Las dos consecuciones anteriores coinciden con dos de los objetivos primordiales de la tesis, la consecución de resultados con alta definición espacial en una cantidad de tiempo factible.

En primer lugar, ha sido necesario testar la validez de la restricción de daño como forma alternativa de incorporar las restricciones de tensión en el problema de optimización topológica, para lo cual se han formulado algunos de los ejemplos que han sido resueltos tradicionalmente en el campo de la optimización topológica de estructuras. A continuación, se han resuelto varios ejemplos bidimensionales en tensión plana de aplicación en el ámbito de la ingeniería. Por último, se han formulado dos problemas en el espacio tridimensional con el objeto de analizar la viabilidad del método desarrollado en esta tesis de cara a la resolución de problemas de optimización topológica de estructuras tridimensionales.

Una vez resueltos los casos propuestos, se puede concluir que el algoritmo desarrollado en esta tesis se ha mostrado robusto y eficaz proporcionando resultados con una alta definición espacial en una cantidad de tiempo aceptable, especialmente en el caso de hacer uso del Análisis Isogeométrico, método que además de proporcionar resultados de mejor calidad para un mismo número de divisiones del dominio, requiere de una menor cantidad de tiempo para resolver el problema cuando el número de esas divisiones aumenta en comparación con el otro método desarrollado.

Por otro lado, la utilización de la restricción de daño como forma de considerar el efecto de las tensiones en el problema de optimización topológica ha demostrado ser una forma alternativa válida a los planteamientos de agrupación de restricciones de tensión desarrollados con anterioridad al proporcionar resultados similares a los obtenidos con el planteamiento de restricciones de tensión local. Además, el uso del método del Estado Adjunto en la obtención del análisis de sensibilidad de la restricción de daño ha sido ventajoso, no solo por ser el número de variables de diseño considerablemente mayor en comparación con el número de restricciones, sino porque ha sido posible aprovechar parte de los resultados obtenidos en el análisis estructural previamente calculado.

El modelo desarrollado en esta tesis abre las puertas a futuras líneas de investigación en el campo de la optimización topológica de estructuras. A continuación, se comentarán aquellas que pueden ser más prometedoras y por tanto tener una mayor relevancia en dicho ámbito.

- La implementación de nuevas formas para definir la distribución de material en el dominio con el objeto de obtener una mayor definición espacial de las soluciones requiriendo de un menor número de divisiones del dominio.
- El desarrollo de la formulación para resolver problemas con grandes deformaciones y grandes desplazamientos.
- El estudio de la posibilidad de utilizar materiales de comportamiento no lineal en la definición del problema.
- El análisis de la viabilidad de la resolución de problemas con geometrías más complejas, no formadas íntegramente solo por rectángulos o prismas en caso de hacer uso del Análisis Isogeométrico.

Por otra parte, también sería interesante realizar la formulación e implementación del análisis de sensibilidad de segundo orden para disponer así de un algoritmo más eficiente, especialmente en lo que respecta al número de iteraciones realizadas para obtener la solución óptima del problema. Por último, también tendría especial interés sobre todo en el caso del Análisis Isogeométrico aplicar las técnicas del cálculo en paralelo al código desarrollado en esta tesis, puesto que dicho desarrollo implicaría una reducción del tiempo de cálculo requerido en el análisis de sensibilidad, lo que a su vez posibilitaría también la resolución de problemas en los que la estructura este sometida a diferentes estados de carga.

Appendix B

Resumo estendido en Galego

"O verdadeiro heroísmo consiste en trocar os anceios en realidades e as ideias en feitos"
Daniel Rodríguez Castelao, (1886-1950).

Introdución

Desde que Bendsøe e Kikuchi establecesen as bases da Optimización Topolóxica en 1988, múltiples contribucións foron desenvoltas no devandito campo. Esas contribucións tiveron como obxecto incrementar o ámbito de aplicación da optimización topolóxica mediante a formulación de diversos problemas que puidesen ser resoltos mediante a aplicación dos seus principios e bases. Iso deu lugar a que a optimización topolóxica poida ser empregada en campos tan diversos como a enxeñería ou a medicina.

No que respecta á optimización topolóxica de estruturas, unha das que máis interese suscita no campo da enxeñería civil, o obxectivo principal é a distribución de material nun certo dominio co obxecto de maximizar ou minimizar unha certa característica do elemento que se está deseñando e garantir á súa vez o cumprimento dunha serie de condicións. As primeiras formulacións do problema de optimización topolóxica de estruturas buscaban obter o deseño de maior rixidez limitando a cantidade de material empregada, estas suscitaron un enorme interese, sendo actualmente as máis utilizadas a pesar de que non era posible asegurar que os elementos deseñados fosen capaces de desenvolver a función para a cal foron concibidos. Iso era debido fundamentalmente a que as cargas ás que estaban sometidos podían provocar tensións inasumibles que ocasionasen a rotura do elemento.

Por mor diso desenvolvéronse formulacións nas que as tensións ou os desprazamentos incorporábanse como restricións no problema de optimización, asegurando así que os devanditos elementos desempeñasen a función para a cal foron deseñados. Doutra banda desenvolveuse a formulación de mínimo peso, onde o obxectivo principal era deseñar estruturas que coa menor cantidade de material posible puidesen desenvolver a súa función, noutras palabras, que sexan capaces de soportar todas as cargas ás que van estar sometidas durante a súa vida útil. Isto último estaba garantido a través da incorporación das restricións en tensión.

As formulacións de mínimo peso con restricións en tensión suscitaron inicialmente un gran interese debido á súa aplicabilidade. Con todo a necesidade de obter solucións con alta definición espacial significou á súa vez a necesidade de dispoñer dun maior número de restricións na definición do problema. Tras o cal e co obxecto de reducir os tempos de cálculo fíxose necesario o desenvolvemento de novas formas de incorporar as restricións de tensión no problema de optimización topolóxica.

Foi así como desenvolvéronse as formulacións de restrición en tensión de tipo global e redución por bloques de restricións de tipo local, onde unha única restrición ten en conta o efecto dun conxunto ou da totalidade dos puntos nos que se comproba a tensión considerados na formulación do problema. Con todo e debido á forma na que as restricións eran formuladas non era posible garantir que todas as tensións fosen inferiores ao seu máximo valor permitido nos resultados obtidos con esta metodoloxía.

Por outra banda, o outro aspecto relevante na formulación do problema de optimización topolóxica de estruturas é a forma na que se define a distribución do material no dominio. Nas primeiras formulacións o dominio era dividido nun certo número de rexións nas cales podía haber ou non material. Noutras palabras, requiríase o uso de variables de deseño discretas. Posteriormente, desenvolveuse a posibilidade de utilizar variables de deseño continuas na formulación do problema, o que permitiu a utilización de algoritmos de optimización na resolución do mesmo mellorando á súa vez a calidade dos resultados obtidos, ao non ser necesario encher completamente cada unha das rexións do dominio, as cales tiñan unha densidade relativa, grao de ocupación da rexión, constante.

Mais adiante e con obxecto de mellorar a calidade dos resultados obtidos e evitar a dependencia da solución con respecto á forma de dividir o dominio, desenvolvéronse formulacións nas que a densidade relativa en cada punto do dominio dependía do valor desta nun certo conxunto de puntos de control ou era directamente calculada a través dunha función matemática que dependía da posición de cada punto dentro do dominio de definición do problema.

Por tanto, é necesaria unha formulación do problema de optimización topolóxica de estruturas de mínimo peso que permita asegurar que todas as tensións estruturais son inferiores a un certo valor máximo mediante o uso dun número reducido de restricións en comparación ao número de variables de deseño empregadas na definición da distribución de material. Á súa vez tamén se pretenden obter resultados cunha alta definición espacial mediante o emprego do menor número de variables de deseño posibles.

E ese é o obxectivo desta tese. Desenvolver unha metodoloxía de optimización topolóxica estrutural que permita a obtención das estruturas de mínimo peso que sexan capaces de soportar todas as cargas ás que van estar sometidas e que proporcione solucións con alta definición espacial mediante o uso dun número razoable de variables de deseño. Por todo iso, os obxectivos específicos da tese son:

- Incorporar a Análise Isoxeométrica na formulación do problema de optimización topolóxica de estruturas como alternativa ao Método dos Elementos Finitos.
- Desenvolver a formulación do problema de optimización topolóxica cunha única restrición que permita considerar o efecto dun gran número de tensións dunha forma efectiva.
- Estender a formulación do problema de optimización topolóxica de estruturas do espazo bidimensional ao espazo tridimensional.
- Lograr que o algoritmo desenvolvido permita resolver o problema de optimización topolóxica nunha cantidade de tempo razoable.
- Resolver diferentes problemas de optimización topolóxica de estruturas non só no espazo bidimensional senón tamén no espazo tridimensional.

Análise estrutural

A análise estrutural do elemento para optimizar calcúlase facendo uso da formulación estándar do Método dos Elementos Finitos para estruturas continuas, dada a natureza continua das tensións e dos desprazamentos estruturais. Para empregar esta formulación consideráronse as seguintes hipóteses: pequenos desprazamentos, pequenas deformacións e material de comportamento elástico lineal. Por outra banda, a utilización da Análise Isoxeométrica non supón unha dificultade inherente dado que o devandito método é derivado do Método dos Elementos Finitos.

Con todo, será necesario ter en conta a influencia das variables de deseño na formulación da análise estrutural, posto que as variables de deseño son usadas para a definición da distribución do material no dominio. Por esa razón, será necesario incorporar as variables de deseño na formulación da matriz de rixidez.

Doutra banda, todos os exemplos bidimensionais expostos nesta tese serán considerados casos de tensión plana. Sendo por tanto necesario incorporar todos os principios da tensión plana na formulación da análise estrutural.

Para o cálculo da análise estrutural ademais das cargas externas que serán aplicadas sobre a estrutura para optimizar tamén será necesario incorporar o valor do peso propio da devandita estrutura, valor que de igual forma que a matriz de rixidez virá condicionado polo valor das variables de deseño. Finalmente, será necesario tamén incorporar o efecto das condicións de contorno usadas na definición do problema xeralmente asociadas a ausencia de desprazamentos en algures do dominio.

Unha vez que a análise estrutural foi calculada obtense como resultado o valor dos desprazamentos en calquera punto do dominio e o valor das reaccións naqueles puntos nos que os desprazamentos foron coartados. Por outra banda, co valor dos desprazamentos en calquera punto do dominio é posible calcular o valor das tensións en calquera punto. Este é o principal obxectivo da análise estrutural desenvolvida, posto que as tensións serán utilizadas para formular a restrición do problema de optimización topolóxica.

A análise estrutural representa a parte máis custosa computacionalmente para unha das formulacións propostas na definición da distribución de material no dominio: densidade uniforme por elemento. Con todo, podería ser posible reducir o custo computacional se se incorporan técnicas máis eficientes no que respecta ao almacenamento de matrices ou renumeración de mallas.

Formulación do problema de optimización

A formulación de calquera problema de optimización matemática require da definición de tres aspectos básicos: a función obxectivo, as restricións e as variables de deseño. Estas últimas son os parámetros que se modifican durante a resolución do problema.

No problema de optimización topolóxica de estruturas formulado nesta tese a función obxectivo correspóndese co peso estrutural, o cal se busca minimizar co obxectivo de reducir os custos de produción e o impacto ambiental que supón a fabricación dun certo compoñente. Neste caso, a relación entre a función obxectivo e as variables de deseño é evidente posto que estas últimas úsanse para definir a distribución do material no dominio.

Ademais e co obxecto de obter solucións baleiro-cheo nas que a maioría das súas variables de deseño sexan iguais aos seus límites inferior ou superior, o cal proporciona solucións máis facilmente manufacturables, procédese a introducir un coeficiente de penalización dos valores intermedios da densidade relativa.

Por outra banda, todo problema de optimización require da existencia de restricións. No caso do problema de optimización exposto nesta tese, buscarase asegurar que as estruturas deseñadas sexan capaces de desempeñar a súa función sen sufrir problema algún. Por esa razón será necesario realizar a comprobación do valor das tensións no dominio estrutural. Para tal fin procédese a introducir unicamente unha restrición baseada no ”Damage Approach” (enfoque do dano) no problema de optimización topolóxica.

Dita restrición será formulada a través da comparación entre dous modelos: un deles é o orixinal e o outro é un modelo danado no cal unha das características do modelo orixinal é perturbada cando o valor das tensións supera o seu máximo valor permitido. Neste caso procédese a incrementar o valor do peso estrutural nas rexións do dominio nas que as tensións superen o seu máximo valor permitido. Por outra banda, o incremento do peso estrutural será maior a medida que o grao de superación da tensión máxima aumente.

Por todo iso, a restrición de dano introducida no problema de optimización topolóxica consistirá nunha comparación do peso total da estrutura de ámbolos modelos. Dado que a restrición de dano é orixinalmente unha restrición de igualdade e por tanto activarase cando o peso de ámbolos modelos sexa diferente, será necesario introducir unha relaxación desta co obxecto de convertela nunha restrición de desigualdade.

Doutra banda, a análise de sensibilidade da restrición non proporcionará ningunha información ata a fronteira entre a rexión factible e non factible ao ser o peso de ámbolos modelos (orixinal e danado) igual en toda a rexión factible, noutras palabras, o valor da restrición de dano vai ser constante e en consecuencia as súas derivadas nulas. Por este motivo, é necesario introducir unha pequena modificación na definición da restrición de dano cando o valor da tensión supere unha certa porcentaxe da máxima tensión permitida que introduza cambios entre o peso do modelo orixinal e o modelo danado.

Con todo, a magnitude das devanditas modificacións poderase considerar despreciable en termos de modificación do peso estrutural pero valiosa á hora de realizar a análise de sensibilidade debido a que se dispoñerá de información para a toma de decisións nas proximidades da fronteira entre a rexión factible e non factible.

Ademais, dado que o material empregado para a definición da estruturas é aceiro, o criterio utilizado na formulación das restricións de tensión que serán consideradas na xeración do modelo danado é o de Von Mises.

Por último, as variables de deseño do problema de optimización topolóxica utilizaranse para definir a distribución do material no dominio. Para tal fin propóñense dúas formulacións alternativas: un baseado nas formulacións clásicas do Método dos Elementos Finitos e o outro mediante a aplicación das bases da Análise Isoxeométrica.

Con respecto ao primeiro método o valor da densidade relativa toma habitualmente un valor constante en cada unha das divisións do dominio e en consecuencia o número de variables de deseño é igual ao número de particións do dominio. Pola contra no segundo método o valor da densidade relativa en calquera punto do dominio ten que ser calculado por medio dunha combinación lineal dos valores da densidade relativa nos puntos de control multiplicado polo valor da función de forma do devandito punto de control no punto onde se quere obter o valor da densidade relativa. Os valores da densidade relativa nos puntos de control son as variables de deseño do problema de optimización polo que o número de variables de deseño é igual ao número de puntos de control, valor que depende das características de definición do modelo de Análise Isoxeométrica considerada.

Por outra banda, quedan por definir os valores límite que poden tomar as variables de deseño. Mentres que o límite superior da variable de deseño é igual á unidade, o que indica ausencia de porosidade, o límite inferior debería ser igual a 0 o que significaría ausencia de material. Con todo, isto non é posible por cuestións numéricas. No caso de permitir a aparición de densidades relativas nulas, a matriz de rixidez obtida na análise estrutural sería singular e en consecuencia devandito análise non tería solución. Por este motivo é necesario establecer un límite inferior das variables de deseño lixeiramente superior a 0. Finalmente, para permitir a aparición de áreas cuxa densidade relativa sexa igual ao seu límite inferior introdúcese unha relaxación das tensións.

Metodoloxía de optimización e análise de sensibilidade

Como norma xeral a propia definición do problema de optimización determina en gran medida a elección do algoritmo matemático de optimización máis adecuado para a resolución do problema. O problema de optimización que se pretende resolver nesta tese consta dunha única función obxectivo, o peso da estrutura; unha única restrición, a restrición de dano; e con variables de deseño continuas no seu dominio ao ser usadas para definir unha variable continua, a densidade relativa do material no dominio de definición da estrutura.

O tipo de problema a resolver nesta tese carece de solución analítica directa. Por esa razón é necesario expor a redución do problema a unha sucesión de problemas máis sinxelos. Noutras palabras, o problema de optimización resólvese por medio dun proceso iterativo. Nesta tese optouse polo mesmo método de redución empregado tipicamente na resolución dos problemas de optimización topolóxica: Programación Lineal Secuencial (SLP).

Aínda que o problema de optimización é non lineal tanto no que respecta á función obxectivo como á restrición de dano, aproxímase a unha secuencia de problemas lineais facendo uso para iso da información obtida na análise de sensibilidade de primeira orde. Por tanto, en cada iteración do proceso de optimización resolverase un problema linealizado mediante a aplicación do algoritmo Simplex.

Con todo, é importante ter en conta que a linealización do problema implica unha perda de información con respecto ao problema de optimización orixinal, o que á súa vez produce unha modificación do problema de optimización a resolver, que terá un dominio factible distinto. Por este motivo e co obxectivo de mitigar as diferenzas entre o problema orixinal e o linealizado introdúcese unha limitación adicional no problema modificado relacionada coa máxima modificación permitida de cada unha das variables de deseño en cada iteración do proceso de optimización. Esta estratexia recibe o nome de límites móbiles, os cales deben ser actualizados en cada iteración tendo en conta non só a máxima modificación permitida se non tamén o valor dos límites laterais impostos na definición do problema de optimización topolóxica: as densidades relativas mínima e máxima.

Por todo iso, para a transformación do problema de optimización orixinal no problema linealizado requírese unicamente da análise de sensibilidade de primeira orde da función obxectivo e da restrición de dano. Mentres que a análise de sensibilidade da función obxectivo pode ser calculada directamente, a análise de sensibilidade da restrición de dano ten que ser calculada facendo uso das técnicas de diferenciación analítica. Neste caso ao ser o número de restricións impostas moito menor que o número de variables de deseño, debido a que só se definiu unha única restrición de dano, procédese a empregar o Método da Variable Adxunta.

A análise de sensibilidade representa a parte máis custosa computacionalmente para unha das formulacións propostas na definición da distribución de material no dominio: a distribución de material por medio dunha interpolación isoxeométrica. Con todo, sería posible reducir o custo computacional se se implementan as técnicas de paralelización no cálculo.

Resultados, conclusións e futuras liñas de investigación

A metodoloxía de optimización topolóxica de estruturas de mínimo peso con restrición de dano desenvolta nesta tese aplicouse a diferentes exemplos tanto para a resolución de problemas bidimensionais en tensión plana como para a resolución de problemas tridimensionais. Con este método obtivéronse por unha banda reducións de tempo de cálculo importantes en comparación cos métodos que incorporaban directamente as restricións de tensión no problema de optimización topolóxica, e doutra banda os resultados obtidos presentan unha elevada definición espacial especialmente no caso da Análise Isoxeométrica. Isto foi posible mediante o incremento do número de variables de deseño usadas na definición da distribución do material no dominio. As dúas consecucións anteriores coinciden con dous dos obxectivos primordiais da tese, a consecución de resultados con alta definición espacial nunha cantidade de tempo factible.

En primeiro lugar, foi necesario testar a validez da restrición de dano como forma alternativa de incorporar as restricións de tensión no problema de optimización topolóxica, para o que se formularon algúns dos exemplos que foron resoltos tradicionalmente no campo da optimización topolóxica de estruturas. A continuación, tense resolto varios exemplos bidimensionais en tensión plana de aplicación no ámbito da enxeñería. Por último, formuláronse dous problemas no espazo tridimensional co obxecto de analizar a viabilidade do método desenvolvido nesta tese para a resolución de problemas de optimización topolóxica de estruturas tridimensionais.

Unha vez resoltos os casos propostos, pódese concluír que o algoritmo desenvolvido nesta tese mostrouse robusto e eficaz proporcionando resultados cunha alta definición espacial nunha cantidade de tempo aceptable, especialmente no caso de facer uso da Análise Isoxeométrica, método que ademais de proporcionar resultados de mellor calidade para un mesmo número de divisións do dominio, require dunha menor cantidade de tempo para resolver o problema cando o número desas divisións aumenta en comparación co outro método desenvolvido.

Doutra banda, a utilización da restrición de dano como forma de considerar o efecto das tensións no problema de optimización topolóxica demostrou ser unha forma alternativa válida ás formulacións de agrupación de restricións de tensión desenvoltos con anterioridade ao proporcionar resultados similares aos obtidos coa formulación de restricións de tensión local. Ademais, o uso do método do Estado Adxunto na obtención da análise de sensibilidade da restrición de dano foi vantaxoso, non só por ser o número de variables de deseño considerablemente maior en comparación co número de restricións, senón porque foi posible aproveitar parte dos resultados obtidos na análise estrutural previamente calculada.

O modelo desenvolvido nesta tese abre as portas a futuras liñas de investigación no campo da optimización topolóxica de estruturas. A continuación, comentaranse aquelas que poden ser máis prometedoras e por tanto ter unha maior relevancia no devandito ámbito.

- A implementación de novas formas para definir a distribución de material no dominio co obxecto de obter unha maior definición espacial das solucións requirindo dun menor número de divisións do dominio.
- O desenvolvemento da formulación para resolver problemas con grandes deformacións e grandes desprazamentos.
- O estudo da posibilidade de utilizar materiais de comportamento non lineal na definición do problema.
- A análise da viabilidade da resolución de problemas con xeometrías máis complexas, non formadas integramente só por rectángulos ou prismas en caso de facer uso da Análise Isoxeométrica.

Por outra banda, tamén sería interesante realizar a formulación e implementación da análise de sensibilidade de segunda orde para dispoñer así dun algoritmo máis eficiente, especialmente no que respecta ao número de iteracións realizadas para obter a solución óptima do problema. Por último, tamén tería especial interese sobre todo no caso da Análise Isoxeométrica aplicar as técnicas do cálculo en paralelo ao código desenvolvido nesta tese, debido a que o devandito desenvolvemento implicaría unha redución do tempo de cálculo requirido na análise de sensibilidade, o que á súa vez posibilitaría tamén a resolución de problemas nos que a estrutura este sometida a diferentes estados de carga.

"Gentlemen, I am ready for the questions to my answers"
Charles de Gaulle, (1890-1970).

Bibliography

Allaire, G., Jouve, F., & Maillot, H. (2004). Topology optimization for minimum stress design with the homogenization method. *Structural and Multidisciplinary Optimization*, (28), 87–98. ↑11

Amstutz, S. & Andrä, H. (2006). A new algorithm for topology optimization using a level-set method. *Journal of Computational Physics*, (216), 573–588. ↑18 , ↑19

Baiges, J., Martínez-Frutos, J., Herrero-Pérez, D., Otero, F., & Ferrer, A. (2019). Large-scale stochastic topology optimization using adaptive mesh refinement and coarsening through a two-level parallelization scheme. *Computer Methods in Applied Mechanics and Engineering*, (343), 186–206. ↑19 , ↑29

Bendsøe, M. P. (1989). Optimal shape design as a material distribution problem. *Structural Optimization*, (1), 193–202. ↑58

Bendsøe, M. P. (1995). *Optimization of Structural Topology, Shape, and Material*. Springer-Verlag. ↑4 , ↑11 , ↑14 , ↑15 , ↑61

Bendsøe, M. P. & Kikuchi, N. (1988). Generating optimal topologies in structural design using a homogenization method. *Computer Methods in Applied Mechanics and Engineering*, (71), 197–224. ↑3 , ↑9 , ↑11 , ↑30 , ↑61

Bendsøe, M. P. & Sigmund, O. (1999). Material interpolation schemes in topology optimization. *Archive of Applied Mechanics*, (69), 635–654. ↑V , ↑14 , ↑15 , ↑16

Blank, L., Garcke, H., Sarbu, L., Srisupattarawanit, T., Styles, V., & Voigt, A. (2012). Phase-field approaches to structural topology optimization. *International Series of Numerical Mathematics*, (160), 245–256. ↑20

Burger, M. & Stainko, R. (2005). Phase-field relaxation of topology optimization with local stress constraints. In *IUTAM Symposium on Topology Design Optimization of Structures Machines and Materials* Rungsted, Denmark. ↑80

Burguer, M., Hackl, B., & Ring, W. (2004). Incorporating topological derivatives into level set methods. *Journal of Computational Physics*, 194, 344–362. ↑29

Burguer, M. & Stainko, R. (2006). Phase-field relaxation of topology optimization with local stress constraints. *SIAM J. Control Optimization*, 48(4), 1447–1466. ↑20

Cao, Y., Li, S., & Petzold, L. (2002). Adjoint sensitivity analysis for differential-algebraic equations: algorithms and software. *Journal of Computational and Applied Mathematics*, (149), 171–191. ↑97

Cheng, G. D. & Jiang, Z. (1992). Study on topology optimization with stress constraints. *Engineering Optimization*, (20), 129–148. ↑56 , ↑80

Chung, S. H., Kwon, Y. S., Park, S. J., & German, R. M. (2009). Sensitivity analysis by the adjoint variable method for optimization of the die compaction process in particulate materials processing. *Finite Elements in Analysis and Design*, (45), 836–844. ↑97

Cox, M. G. (1972). The numerical evaluation of b-splines. *IMA Journal of Applied Mathematics*, 10(2), 134–149. ↑36

Dantzig, G. B. & Thapa, M. N. (1997). *Linear Programming I: Introduction.* Springer-Verlag. ↑84

Dantzig, G. B. & Thapa, M. N. (2003). *Linear Programming II: Theory and extensions.* Springer-Verlag. ↑84

de Boor, C. (1972). On calculating with b-splines. *Journal of Approximation Theory*, 6, 50–62. ↑36

Dedè, L., Borden, M. J., & Hughes, T. J. R. (2012). Isogeometric analysis for topology optimization with a phase field model. *Archives of Computational Methods in Engineering*, (19), 427–465. ↑V , ↑20 , ↑22

Dunning, P. D. & Kim, H. A. (2013). A new hole insertion method for level set based structural topological optimization. *International Journal for Numerical Methods in Engineering*, (93), 118–134. ↑19 , ↑20

Duysinx, P. (1998). *Topology optimization with different stress limits in tension and compression, International report: Robotics and Automation.* Institute of Mechanics, University of Liege. ↑80

Duysinx, P. & Bendsøe, M. P. (1998). Topology optimization of continuum structures with local stress constraints. *International Journal of Numerical Methods in Engineering*, (43), 1453–1478. ↑23 , ↑80

Duysinx, P. & Sigmund, O. (1998). New development in handling stress constraints in optimal material distribution. In *7th AIAA/USAF/NASA/ISSMO Symposium on Multidisciplinary Design Optimization* Missouri, USA. ↑25 , ↑56 , ↑80

Eschenauer, H. A., Kobelev, V. V., & Schumacher, A. (1994). Bubble method for topology and shape optimization of structures. *Structural Optimization*, (8), 42–51. ↑V , ↑20

Eschenauer, H. A. & Olhoff, N. (2001). Topology optimization of continuum structures: A review. *Applied Mechanics Review*, (54), 331–389. ↑V , ↑4 , ↑11 , ↑15

Eschenauer, H. A. & Schumacher, A. (1993). Bubble method: a special strategy for finding best possible initial designs. *Proceedings of the ASME Design Technical Conference - 19th Design Automotion Conference*, (65), 437–443. ↑19

Ferrer, A. (2017). *Multi-scale topological design of structural materials.* PhD thesis, Polytechnic University of Cataluña. ↑29

Gao, J., Luo, Z., Li, H., & Gao, L. (2019). Topology optimization for multiscale design of porous composites with multi-domain microstructures. *Computer Methods in Applied Mechanics and Engineering*, (344), 451–476. ↑17

Giusti, S. M., Novotny, A. A., & Padra, C. (2008). Topological sensitivity analysis of inclusion in two-dimensional linear elasticity. *Engineering Analysis with Boundary Elements*, (32), 926–935. ↑29

Hassani, B., Khanzadi, M., & Tavakkoli, S. M. (2012). An isogeometrical approach to structural topology optimization by optimality criteria. *Structural Multidisciplinary Optimization*, (45), 223–233. ↑22 , ↑23

Hughes, T. (2000). *The finite element method: Linear static and dynamic finite element analysis*, (pp. 109–181). Dover Publications. ↑36 , ↑50 , ↑51 , ↑54

Hughes, T., Cottrell, J., & Bazilevs, Y. (2005). Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement. *Computer Methods in Applied Mechanics and Engineering*, (194), 4135–4195. ↑V , ↑22 , ↑37

Hughes, T., Cottrell, J. A., & Bazilevs, Y. (2009). *Isogeometric Analysis: Toward Integration of* CAD *and* FEA. Wiley Publication. ↑22

Jeong, S. H., Yoon, G. H., Takezawa, A., & Choi, D.-H. (2014). Development of a novel phase-field method for local stress-based shape and topology optimization. *Computers and Structures*, (132), 84–98. ↑20

Jr., H. E. & Fancello, E. A. (2014). A level set approach for topology optimization with local stress constraints. *International Journal for Numerical Methods in Engineering*, (99), 129–156. ↑19

Kennedy, G. J. & Hicken, J. E. (2015). Improved constraint-aggregation methods. *Computational Methods in Applied Mechanics and Engineering*, (289), 332–354. ↑25 , ↑26

Kim, Y. Y. & Yoon, G. H. (2000). Multi-resolution multi-scale topology optimization - a new paradigm. *International Journal of Solids and Structures*, (37), 5529–5559. ↑27

Kohn, R. V. & Strang, G. (1986a). Optimal design and relaxation of variational problems. *Communications of Pure and Applied Mathematics*, Part I(39), 1–25. ↑11

Kohn, R. V. & Strang, G. (1986b). Optimal design and relaxation of variational problems. *Communications of Pure and Applied Mathematics*, Part II(39), 139–182. ↑11

Kohn, R. V. & Strang, G. (1986c). Optimal design and relaxation of variational problems. *Communications of Pure and Applied Mathematics*, Part III(39), 353–377. ↑11

Kreisselmeier, G. & Steinhauser, R. (1979). Systematic control design by optimizing a vector performance indicator. *Symposium on Computer-Aided Design of Control Systems*, (1), 113–117. ↑25

Kreisselmeier, G. & Steinhauser, R. (1983). Application of vector performance optimization to a robust control loop design for a fighter aircraft. *International Journal Control*, 2(37), 251–284. ↑25

Lambe, A. B., Kennedy, G. J., & Martins, J. R. R. A. (2017). An evaluation of constraint aggregation strategies for wing box mass minimization. *Structural Multidisciplinary Optimization*, (55), 257–277. ↑25

Lee, T. H. (1999). An adjoint variable method for structural design sensitivity analysis of a distinct eigenvalue problem. *Journal of Mechanical Science and Technology*, (13), 470–476. ↑97

Lieu, Q. X. & Lee, J. (2017). A multi-resolution approach for multi-material topology optimization based on isogeometric analysis. *Computer Methods in Applied Mechanics and Engineering*, (323), 272–302. ↑23

Liu, H., Yang, D., Hao, P., & Zhu, X. (2018). Isogeometric analysis based topology optimization design with global stress constraint. *Computer Methods in Applied Mechanics and Engineering*, (342), 625–652. ↑23 , ↑34

Liu, Z. & Korvink, J. G. (2008). Adaptive moving mesh level set method for structure topology optimization. *Engineering Optimization*, 40(6), 529–558. ↑18 , ↑19

Lurie, K. & Cherkaev, A. (1997). Effective characteristics of composite materials and the optimal design of structural elements. *Uspekhi Mekh.*, (57), 91–100. ↑11

Martins, J. R. R. A. & Poon, N. M. K. (2005). On structural optimization using constraint aggregation. In *VI World Congress on Structural and Multidisciplinary Optimization* Rio de Janeiro, Brazil. ↑24

Martínez-Frutos, J., Allaire, G., Dapogny, C., & Periago, F. (2019). Structural optimization under internal porosity constraints using topological derivative. *Computer Methods in Applied Mechanics and Engineering*, (345), 1–25. ↑29

Ministerio de Vivienda (2006). *Código Técnico de la Edificación.* Ministerio de Vivienda. ↑79

Muiños, I. (2001). *Optimización Topológica de Estructuras: Una Formulación de Elements Finitos para la Minimización del Peso con Restricciones en Tensión.* Technical report, ETSICCP, University of A Coruña. ↑80

Muiños, I., Colominas, I., Navarrina, F., & Casteleiro, M. (2002). Una formulación de mínimo peso con restricciones en tensión para la optimización topológica de estructuras. *Métodos Numéricos en Ingeniería y Ciencias Aplicadas*, (pp. 399–408). ↑23

Murat, F. & Tartar, L. (1985). Calcul des variations e homogénéisation, les méthodes de l' homogénéisation théorie et applications en physique. *Collegue Dir. Etudes et Recherches EDF*, (57), 319–369. ↑11 , ↑12

Nakshatrala, P. B., Tortorelli, D. A., & Nakshatrala, K. (2013). Nonlinear structural design using multiscale topology optimization. part i: Static formulation. *Computer Methods in Applied Mechanics and Engineering*, (261-262), 167–176. ↑16

Navarrina, F. (1987). *Una metodología general para la optimización estructural en diseño asistido por ordenador.* PhD thesis, Polytechnic University of Cataluña. ↑84

Navarrina, F. & Casteleiro, M. (1991). A general methodological analysis for optimum design. *International Journal of Numerical Methods in Engineering*, (31), 85–111. ↑84

Navarrina, F., López, S., Colominas, I., Bendito, E., & Casteleiro, M. (2000). High order shape design sensitivity: A unified approach. *Computer Methods in Applied Mechanics and Engineering*, 188(4), 681–696. ↑94

Navarrina, F., Muiños, I., Colominas, I., & Casteleiro, M. (2002). Optimización topológica de estructuras: Una formulación de mínimo peso con restricciones en tensión. *Métodos Numéricos en Ingeniería V*, (pp. 1–17). ↑23 , ↑80

Navarrina, F., Muiños, I., Colominas, I., & Casteleiro, M. (2003). Minimum weight with stress constraints topology optimization. In *Proceedings del 7th US National Congress on Computational Mechanics* Dearborn, USA. ↑23

Navarrina, F., Muiños, I., Colominas, I., & Casteleiro, M. (2005). Topology optimization of structures: a minimum weight approach with stress constraints. *Advances in Engineering Software*, (36), 599–606. ↑50 , ↑51 , ↑54 , ↑58 , ↑80

Navarrina, F., Tarrech, R., Colominas, I., Mosqueira, G., Gómez-Calviño, J., & Casteleiro, M. (2001a). Efficient structural shape optimization: using directional high order sensitivity analysis to improve mp algorithms. In *Proceedings del 6th US National Congress on Computational Mechanics* Dearborn, USA. ↑84

Navarrina, F., Tarrech, R., Colominas, I., Mosqueira, G., Gópez-Calviño, J., & Casteleiro, M. (2001b). An efficient MP algorithm for structural shape optimization problems. *Computer Aided Optimum Design of Structures VII*, (pp. 247–256). ↑84

Nguyen, T. H., Paulino, G. H., Song, J., & Le, C. H. (2010). A computational paradigm for multiresolution topology optimization (mtop). *Structural Multidisciplinary Optimization*, (41), 525–539. ↑V , ↑27 , ↑28

Nguyen, T. H., Paulino, G. H., Song, J., & Le, C. H. (2012). Improving multiresolution topology optimization via multiple discretizations. *International Journal for Numerical Methods In Engineering*, (92), 507–530. ↑V , ↑28

Novotny, A. A., Feijóo, R. A., Taroco, E., & Padra, C. (2007). Topological sensitivity analysis for three-dimensional linear elasticity problem. *Computational Methods Applied Mechanics and Engineering*, (196), 4354–4364. ↑29

Novotny, A. A., Lopes, C., & dos Santos, R. B. (2015). Topological derivative-based topology optimization of structures subject to multiple load-cases. *Latin American Journal of Solids and Structures*, 12(5), 834–860. ↑29

Oñate, E. (1995). *Cálculo de Estructuras por el Método de elementos Finitos: Análisis Estático y Lineal*. CIMNE. ↑50 , ↑54

Olhoff, N. & Eschenauer, H. A. (1999). On optimum topology design in mechanics. In *Proceedings of the European Conference on Computational Mechanics* Munich, Germany. ↑V , ↑12 , ↑13

Park, J. & Sutradhar, A. (2015). A multi-resolution method for 3d multi-material topology optimization. *Computer Methods in Applied Mechanics and Engineering*, (285), 571–586. ↑28

Park, K.-S. & Youn, S.-K. (2005). Globally convergent topology optimization using level set method. In *6th World Congresses of Structural and Multidisciplinary Optimization* Rio de Janeiro, Brazil. ↑18

París, J. (2007). *Restricciones en tensión y minimización del peso: Una metodología general para la optimización topologica de estructuras*. PhD thesis, University of A Coruña. ↑VI , ↑25 , ↑26 , ↑58 , ↑64 , ↑65 , ↑80 , ↑81 , ↑84 , ↑87 , ↑90 , ↑97

París, J., Navarrina, F., Colominas, I., & Casteleiro, M. (2007a). Agrupación de restricciones en tensión por bloques en optimización topológica de estructuras continuas. In *CMNE/CILAMCE 2007* Porto, Portugal. ↑24

París, J., Navarrina, F., Colominas, I., & Casteleiro, M. (2007b). Block aggregation of stress constraints in topology optimization of structures. In *10th International Conference on Computer Aided Optimum Design in Engineering - OPTI 2007* Myrtle Beach, USA. ↑24

París, J., Navarrina, F., Colominas, I., & Casteleiro, M. (2007c). Global versus local statement of stress constraints in topology optimization of continuum structures. In *10th International Conference on Computer Aided Optimum Design in Engineering - OPTI 2007* Myrtle Beach, USA. ↑24

París, J., Navarrina, F., Colominas, I., & Casteleiro, M. (2008). Advances in the statement of stress constraints in structural topology optimization. In *4th International Conference on Advanced Computational Methods in Engineering - ACOMEN 2008* Liège, Belgium. ↑24

París, J., Navarrina, F., Colominas, I., & Casteleiro, M. (2010). Improvements in the treatment of stress constraints in structural topology optimization problems. *Journal of Computational and Applied Mathematics*, (234), 2231–2238. ↑24

Pereira, J. T., Barcellos, C. S., & Fancello, E. A. (2004). Topology optimization of continuum structures with material failure constraints. *Structural and Multidisciplinary Optimization*, (26), 50–66. ↑56 , ↑80

Poon, N. M. K. & Martins, J. R. R. A. (2007). An adaptive approach to constraint aggregation using adjoint sensitivity analysis. *Structural Multidisciplinary Optimization*, (34), 61–73. ↑24

Qian, X. (2013). Topology optimization in b-spline space. *Computer Methods in Applied Mechanics and Engineering*, (265), 15–35. ↑23

Qiu, G. Y. & Liu, X. S. (2010). A note on the derivation of global stress constraints. *Structural Multidisciplinary Optimization*, (40), 625–628. ↑25

Rodrigues, H., Guedes, J. M., & Bends∅e, M. P. (2002). Hierarchical optimization of material and structure. *Structural Multidisciplinary Optimization*, (24), 1–10. ↑V , ↑16 , ↑17

Rogers, D. (2001). *An introduction to NURBS with historical perspective.* Academic Press, San Diego, CA. ↑33

Roodsarabi, M., Khatibinia, M., & Sarafrazi, S. R. (2016). Isogeometric topology optimization of structures using level set method incorporating sensitivity analysis. *International Journal of Optimization in Civil Engineering*, 3(6), 405–422. ↑19

Rossow, H. P. & Taylor, J. E. (1973). A finite element method for the optimal design of variable thickness sheets. *AIAA Journal*, (11), 1566–1569. ↑13

Schumacher, A. (1995). *Topologieoptimierung von bauteilstrukturen unter verwendung von lochpositionierungkriterien.* PhD thesis, Universität-Gesamthochschule-Siegen. ↑29

Sethian, J. A. & Wiegmann, A. (2000). Structural boundary design via level set and immersed interface methods. *Journal of Computational Physics*, (163), 489–528. ↑18

Shojaee, S., Mohamadian, M., & Valizadeh, N. (2012). Composition of isogeometric analysis with level set method for structural topology optimization. *International Journal of Optimization in Civil Engineering*, 1(2), 47–70. ↑19

Sigmund, O. (1994). *Design of material structures using topology optimization.* PhD thesis, Department of Solid Mechanics, DTU, Denmark. ↑56

Sivapuram, R., Dunning, P. D., & Kim, H. A. (2016). Simultaneous material and structural optimization by multiscale topology optimization. *Structural Multidisciplinary Optimization*, (54), 1267–1281. ↑16

Sokolowski, J. & Zochowski, Z. (1999). The topological derivative method in shape optimization. *SIAM Journal on Control and Optimization*, 37(4), 1251–1272. ↑29

Suzuki, K. & Kikuchi, N. (1991). A homogeneization method for shape and topology optimization. *Computer Methods in Applied Mechanics and Engineering*, 93(3), 291–318. ↑11

van Dijk, N. P., Maute, K., Langelaar, M., & van Keulen, F. (2013). Level-set methods for structural topology optimization: a review. *Structural Multidisciplinary Optimization*, (48), 437–472. ↑19

Verbart, A., Langelaar, M., & van Keulen, F. (2016). Damage approach: A new method for topology optimization with local stress constraints. *Structural Multidisciplinary Optimization*, (53), 1081–1098. ↑29 , ↑30 , ↑67 , ↑68

Wang, M. Y., Wang, X., & Guo, D. (2003). A level set method for structural topology optimization. *Computer Methods in Applied Mechanics and Engineering*, (192), 227–246. ↑18

Wang, M. Y. & Zhou, S. (2004). Phase field: A variational method for structural topology optimization. *Tech Science*, 6(6), 547–566. ↑V , ↑20 , ↑21

Wang, Y. & Benson, D. J. (2016). Isogeometric analysis for parameterized lsm-based structural topology optimization. *Computational Mechanics*, (57), 19–35. ↑19

Wang, Y., Xu, H., & Pasini, D. (2017). Multiscale isogeometric topology optimization for lattice materials. *Computer Methods in Applied Mechanics and Engineering*, (316), 568–585. ↑23

Xia, L. & Breitkopf, P. (2014). Current topology optimization design of material and structure within fe^2 non linear multiscale analysis framework. *Computer Methods in Applied Mechanics and Engineering*, (278), 524–542. ↑16

Zhang, Y., Xiao, M., Li, H., Gao, L., & Chu, S. (2018). Multiscale concurrent topology optimization for cellular structures with multiple microstructures based on ordered simp interpolation. *Computational Materials Science*, (155), 74–91. ↑17

Zienkiewickz, O. C. & Taylor, R. L. (2004). *El método de los elementos finitos, Vol. 1. y 2.* CIMNE. ↑50 , ↑51 , ↑54